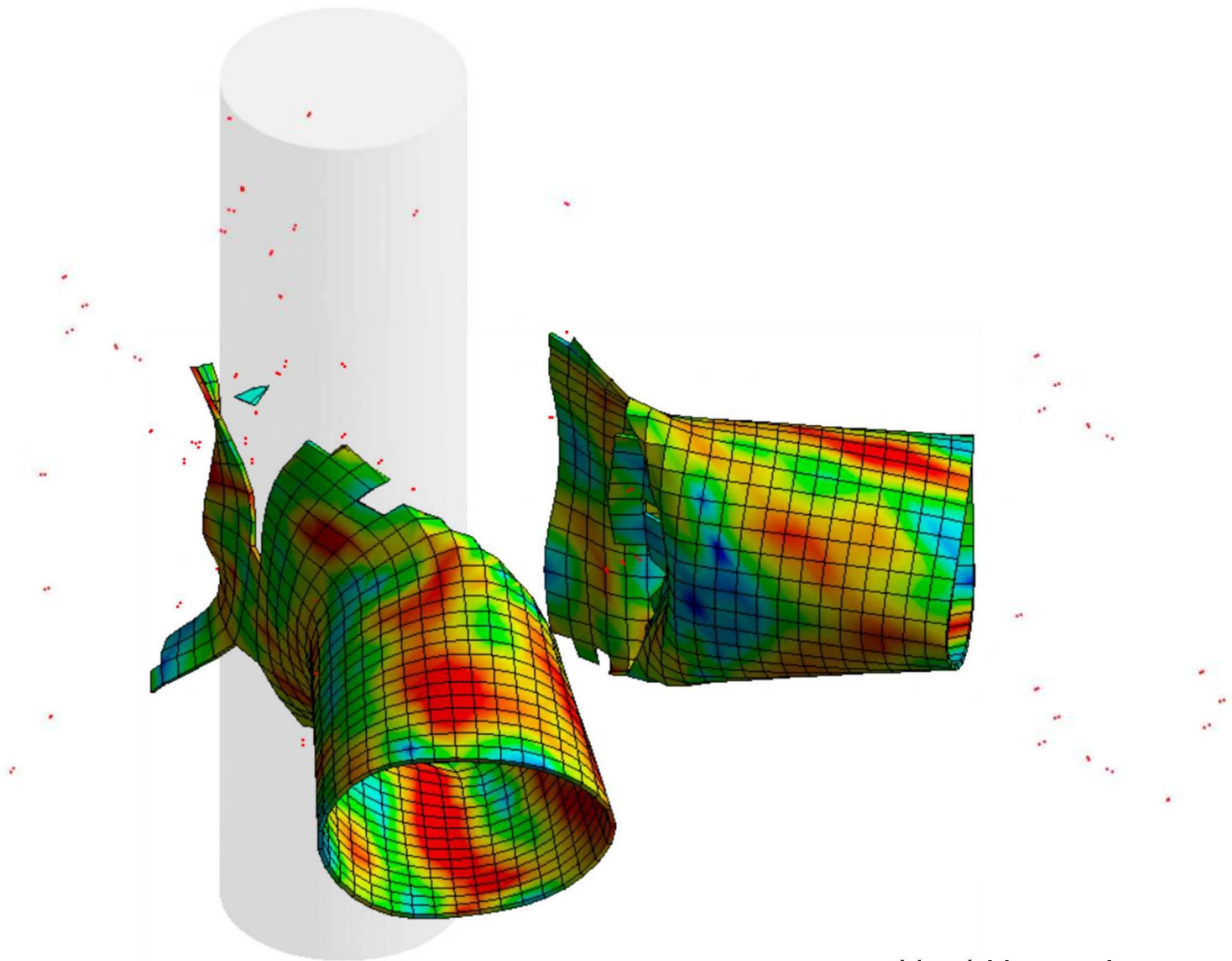


Finite Element Simulations

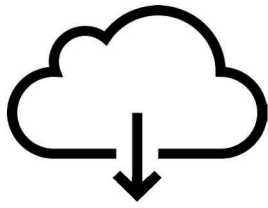
with ANSYS® Workbench 2021

Theory, Applications, Case Studies

Printed
in Full Color



Huei-Huang Lee



REDEEM YOUR CODE

To access exclusive bonus content

This book comes with a unique access code that gives you access to exclusive bonus content. Please email your proof of purchase to service@SDCpublications.com in order to receive your access code.

Once you've redeemed your code the book will be added to your SDC Publications Library. You can access your files wherever and whenever you want by logging into your account and going to your account library.

REDEEM YOUR CODE

1. Email your ebook proof of purchase to service@sdcpublications.com
2. Login to your SDC Publications account or register at SDCpublications.com/Register
3. Once you are logged in to your account visit SDCpublications.com/Redeem
4. Enter the code you received
5. Go to SDCpublications.com/Library to access this book's exclusive content from your account library.

For answers to frequently asked questions regarding downloads and opening files visit SDCpublications.com/FAQ

For technical support visit SDCpublications.com/Contact-Us

Instructors: You can access this book's downloads by logging into your instructor account and visiting this book's details page on our website.

Finite Element Simulations with ANSYS Workbench 2021

Huei-Huang Lee

Department of Engineering Science
National Cheng Kung University, Taiwan



SDC Publications

P.O. Box 1334

Mission, KS 66222

913-262-2664

www.SDCpublications.com

Publisher: Stephen Schroff

Copyright 2021 Huei-Huang Lee

All rights reserved. This document may not be copied, photocopied, reproduced, transmitted, or translated in any form or for any purpose without the express written consent of the publisher, SDC Publications.

It is a violation of United States copyright laws to make copies in any form or media of the contents of this book for commercial or educational purposes without written permission.

Examination Copies

Books received as examination copies are for review purposes only and may not be made available for student use. Resale of examination copies is prohibited.

Electronic Files

Any electronic files associated with this book are licensed to the original user only. These files may not be transferred to any other party.

Trademarks:

ANSYS, ANSYS Mechanical, ANSYS Multiphysics, Workbench, and any and all ANSYS, Inc. product and service names are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. All other trademarks or registered trademarks are the property of their respective owners.

ISBN-13: 978-1-63057-456-7

ISBN-10: 1-63057-456-2

Printed and bound in the United States of America.

Contents

Preface 4

Chapter 1 Introduction 9

- 1.1 Case Study: Pneumatically Actuated PDMS Fingers 10
- 1.2 Structural Mechanics: A Quick Review 24
- 1.3 Finite Element Methods: A Concise Introduction 35
- 1.4 Failure Criteria of Materials 40
- 1.5 Review 47
- 1.6 Appendix: An Unofficial History of ANSYS 51

Chapter 2 Sketching 56

- 2.1 W16x50 Beam 57
- 2.2 Triangular Plate 70
- 2.3 More Details 81
- 2.4 M20x2.5 Threaded Bolt 92
- 2.5 Spur Gears 97
- 2.6 Microgripper 103
- 2.7 Review 107

Chapter 3 2D Simulations 109

- 3.1 Triangular Plate 110
- 3.2 Threaded Bolt-and-Nut 125
- 3.3 More Details 140
- 3.4 Spur Gears 151
- 3.5 Structural Error, FE Convergence, and Stress Singularity 157
- 3.6 Review 170

Chapter 4 3D Solid Modeling 172

- 4.1 Beam Bracket 173
- 4.2 Cover of Pressure Cylinder 179
- 4.3 Lifting Fork 190
- 4.4 More Details 197
- 4.5 LCD Display Support 203
- 4.6 Review 207

Chapter 5 3D Simulations 209

- 5.1 Beam Bracket 210
- 5.2 Cover of Pressure Cylinder 219
- 5.3 More Details 227
- 5.4 LCD Display Support 231
- 5.5 Review 236

Chapter 6 Surface Models 238

- 6.1 Bellows Joints 239
- 6.2 Beam Bracket 249
- 6.3 Gearbox 256
- 6.4 Review 269

Chapter 7 Line Models 271

- 7.1 Flexible Gripper 272
- 7.2 3D Truss 283
- 7.3 Two-Story Building 295
- 7.4 Review 309

Chapter 8 Optimization 311

- 8.1 Flexible Gripper 312
- 8.2 Triangular Plate 323
- 8.3 Review 329

Chapter 9 Meshing 330

- 9.1 Pneumatic Fingers 331
- 9.2 Cover of Pressure Cylinder 346
- 9.3 Convergence Study of 3D Solid Elements 353
- 9.4 Review 365

Chapter 10 Buckling and Stress Stiffening 367

- 10.1 Stress Stiffening 368
- 10.2 3D Truss 379
- 10.3 Beam Bracket 383
- 10.4 Review 387

Chapter 11 Modal Analysis 389

- 11.1 Gearbox 390
- 11.2 Two-Story Building 395
- 11.3 Compact Disk 402
- 11.4 Guitar String 410
- 11.5 Review 417

Chapter 12 Transient Structural Simulations 419

- 12.1 Basics of Structural Dynamics 420
- 12.2 Lifting Fork 429
- 12.3 Harmonic Response Analysis: Two-Story Building 441
- 12.4 Disk and Block 448
- 12.5 Guitar String 456
- 12.6 Review 466

Chapter 13 Nonlinear Simulations 468

- 13.1 Basics of Nonlinear Simulations 469
- 13.2 Translational Joint 481
- 13.3 Microgripper 495
- 13.4 Snap Lock 508
- 13.5 Review 524

Chapter 14 Nonlinear Materials 526

- 14.1 Basics of Nonlinear Materials 527
- 14.2 Belleville Washer 536
- 14.3 Planar Seal 553
- 14.4 Review 568

Chapter 15 Explicit Dynamics 570

- 15.1 Basics of Explicit Dynamics 571
- 15.2 High-Speed Impact 577
- 15.3 Drop Test 587
- 15.4 Review 599

Index 601

Preface

Who is this Book for?

This book is developed mainly for graduate and senior undergraduate students. It may be used in courses such as Computer-Aided Engineering (CAE) or Finite Element Analysis (FEA).

Why ANSYS?

ANSYS is almost a synonym of finite element simulations. I've been using ANSYS as a teaching platform for more than 30 years. I prefer ANSYS to other CAE software because of its multiphysics capabilities, completeness of on-line documentation, and popularity among both academia and industry. Equipping engineering students with multiphysics capabilities is becoming a necessity. Complete documentation allows students to advance further after taking an introductory CAE course. Popularity among academia and industry implies that an engineering student, after his graduation from college, can work with the software without any further training. In recent years, I have additional reasons to advocate this software: user-friendly **Workbench** and **free student products**.

Workbench

For many years, I had been using ANSYS Classic, nowadays dubbed Mechanical APDL. The unfriendly APDL imposes unnecessary constraints, making the software difficult to use. As a result, students or engineers often limit themselves to certain applications; for example, working on component simulations rather than assembly simulations. Workbench adds friendliness on top of the power of APDL, releasing many unnecessary constraints.

Free Student Products

Starting from Release 17, ANSYS provides a free product license for students anywhere in the world. The only limitation is the problem size, which should be less than 32,000 nodes/elements (for shell elements, the limitation is 16,000 nodes/elements) for structural physics. All examples in this book are designed to meet this limitation and tested with the free student version. The free student product can be downloaded from the following ANSYS webpage:

<http://www.ansys.com/Products/Academic/ANSYS-Student>

Why a New Tutorial?

Preparing a tutorial for the Workbench needs much more effort than that for the APDL, due to the graphic nature of the interface. The most comprehensive tutorials, to my knowledge so far, are the training tutorials prepared by ANSYS Inc. However, they may not be suitable for use as a college tutorial for the following reasons. First, the cases used are either too complicated or too trivial. Many of the cases are geometrically too complicated for students to create from scratch. The students need to rely on the geometry files that come with the tutorials. Students usually obtain a better comprehension by working from scratch. Second, the tutorials cover too little on theories of finite element methods and solid mechanics while too much on software operation details. We try to provide a more suitable tutorial for use as a college textbook.

Organization of the Book

We'll describe the organization of the book in Section 1.1. Here is a quick overview of the book.

Using a case study, Section 1.1 walks through a typical Workbench simulation procedure. As more concepts or tools are needed, specific chapters or sections are pointed out, in which an in-depth discussion will be provided. The rest of Chapter 1 provides introductory background essential for the discussion in later chapters. This background includes a concise knowledge of structural mechanics: equations that govern the behavior of a mechanical or structural system (Section 1.2), the finite element methods that solve these governing equations (Section 1.3), and the failure criteria of materials (Section 1.4). Chapter 1 is the only chapter that doesn't have any hands-on exercises because, in the very beginning of a semester, students may not be ready to access the software facilities.

Starting from Chapter 2, a learning-by-doing approach is used throughout the book. Chapters 2 and 3 cover 2D geometric modeling and simulations. Chapters 4-7 introduce 3D geometric modeling and simulations. The next two chapters are dedicated to two useful topics: Chapter 8 to optimization and Chapter 9 to meshing. Chapter 10 deals with buckling and its related topic: stress stiffening. Chapters 11 and 12 discuss dynamic simulations: Chapter 11 is on modal analysis while Chapter 12 is on transient structural analysis. Up to this point, the discussions are mostly on linear problems. Although several nonlinear simulations have been performed, their nonlinear behaviors were not discussed further. Chapters 13 and 14 discuss nonlinear simulations in a more in-depth way. Chapter 15 introduces an exciting topic: explicit dynamics, which is becoming a necessary discipline for a CAE engineer.

Features of the Book

To be **Comprehensive and friendly** is the ultimate goal of this book. To achieve this goal, the following features are incorporated in the book.

Real-World Cases. There are 45 step-by-step hands-on exercises in the book, each completed in a single section. These exercises are designed from 27 real-world cases, carefully chosen to ensure that they are neither too trivial nor too complicated. Many of them are industrial or research projects; pictures of real-world products are presented whenever available. The size of the problems are not too large so that they can be performed in a free ANSYS Student Product, which has a limitation of 32,000 nodes/elements. They are not too complicated so that the students can build each project step by step from scratch themselves. Throughout the book, the students don't need any supplementary files to work on these exercises.

Theoretical Background. Relevant background knowledge is provided whenever necessary, such as solid mechanics, finite element methods, structural dynamics, nonlinear solution methods (e.g., Newton-Raphson methods), nonlinear materials (e.g., plasticity, hyperelasticity), implicit and explicit integration methods, etc. To be efficient, the teaching methods are conceptual rather than mathematical; concise, yet comprehensive. The last four chapters (chapters 12-15) cover more advanced topics, each chapter having an opening section that gives basics of that topic in an efficient way to facilitate the subsequent learning.

How the Workbench internally solves a model is also illustrated throughout the book. Understanding these procedures, at least conceptually, is useful for a CAE engineer.

Key concepts are inserted whenever necessary. Must-know concepts, such as structural error, finite element convergence, and stress singularity, are taught by using designed hands-on exercises, rather than by abstract lecturing. For example, how finite element solutions converge to their analytical solutions, as the meshes become finer, is illustrated by guiding the students to plot convergence curves. That way, the students should have strong knowledge of the finite elements convergence behaviors. And, after hours of laborious working, they will not forget it for the rest of their life! Step-by-step procedures guiding the students to plot curves to illustrate important concepts are one of the featured teaching methods in this book.

Learning by Hands-on Exercises. A learning approach emphasizing hands-on exercises is used through the entire book. In my own experience, this is the best way to learn a complicated software such as ANSYS Workbench. A typical chapter, such as Chapter 3, consists of 6 sections, the first two sections providing two step-by-step examples, the third section giving a more systematic view of the chapter subjects, the following two sections providing more exercises in a not-so-step-by-step way, and the final section providing review problems.

Demo Videos. Each of the 45 step-by-step exercises has been screen-recorded. See Access Code or Author's Webpage, next page, for details.

ANSYS On-line References. One of the objectives of this book is to serve as a guide through the huge repository of ANSYS on-line documentation, a well of knowledge for many students and engineers. The on-line documentation includes a theory reference, an element reference, and many examples. Whenever helpful, we point to a location in the on-line documentation as further study for the students.

End-of-Chapter Keywords. Keywords are summarized at the ending section of each chapter in a quiz form. One goal of this book is to help the students comprehend the terminology and use it efficiently. That is not always easy for some students. For example, whenever asked "What are shape functions?" most of the students cannot satisfyingly define the terminology. Yes, many textbooks spend pages teaching students what the shape functions are, but the challenge is how to define or describe a term in less than two lines of words. This part of the book demonstrates how to define or describe a term in an efficient way; for example, "Shape functions serve as interpolating functions, to calculate continuous displacement fields from discrete nodal displacements."

To Instructors: How I Use the Book

I use this book in a 3-credit 18-week course, Computer-Aided Engineering. The progress is one chapter per week, except Chapter I, which takes 2 weeks. Each week, after a classroom introduction of a chapter using lecture slides, I set up a discussion forum in an e-learning system maintained by the university. After completing the exercises of the chapter, the students are required to participate in the discussion forum.

In the forum, the students may post their questions, help or answer other students' questions, or share their comments. In addition to taking part in the discussion, I rate each posted article with 0-5 stars; the sum of the ratings becomes the grade of a student's performance for the week. The weekly discussion closes before next classroom hours.

The course load is not light. Nevertheless, most of the students were willing to spend hours working on these step-by-step exercises, because these exercises are tangible, rather than abstract. Students of this generation are usually better in picking up knowledge through tangible software exercises rather than abstract lecturing. Further, the students not only feel comfortable to post questions in the forum but also enjoy helping other students or sharing their comments with others.

At the end of the semester, each student is required to turn in a project. Students who are currently working as engineers usually choose topics related to their jobs. Students who are working on their theses usually choose topics related to their studies. They are also allowed to repeat a project found in any reference, as long as they go through all details by themselves. The purpose of the final project is to ensure that students are capable of carrying out a project independently, which is the goal of the course, not just following the step-by-step procedures in the book.

To Students: How My Students Use the Book

Many students, when following the steps in the book, often made mistakes and ended up with different results from those in the book. In many cases they cannot figure out in which steps the mistakes were made. In that case, they have to redo the exercise from the beginning. It is not uncommon that they redid the exercise several times and finally saw the beautiful, reasonable results.

What I want to say is that you may come across the same situation, but you are not wasting your time when you redo the exercises. You are learning from the mistakes. Each time you fix a mistake, you gain more insight. After you obtain the reasonable results, redo it and try to figure out if there are other ways to accomplish the same results. That's how I learned finite element simulations when I was a young engineer.

Finite element methods and solid mechanics are the foundation of mechanical or structural simulations. If you haven't taken these courses, plan to take them after you complete this course. If you've already taken them and still feel uncomfortable, review them.

Project files

Finished project files are available. See Access Code or Author's Webpage below for details.

If everything works smoothly, you don't need the finished project files at all. Every project can be built from scratch according to the steps described in the book. The author provides these project files just in case you need them. For example, when you run into troubles and you don't want to redo from the beginning, you may find these files useful. Or, when you have trouble following the geometry details in the book, you may need to look up the geometry details from the project files.

The most important reason we provide the finished project files is as follows. It is strongly suggested that, in the beginning of an exercise when previously saved project files are needed, you use the project files provided by the author rather than your own files so that you are able to obtain results that have minimum differences in numerical values from those in the textbook.

These finished project files are saved in the ZIP compressed format. Please decompress a file before it is opened with ANSYS Workbench.

Access Code

Each copy of this book includes an access code which gives you access to the finished project files, demonstration videos, and lecture slides. Instructions to redeem your code are located on the inside of the front cover of this book.

Author's Webpage

A webpage dedicated to this book is maintained by the author. The project files, demonstration videos, and lecture slides can be downloaded from the webpage:

<http://lab.es.ncku.edu.tw/hhlee/Site/ANSYS2021.html>

Notations

To efficiently present the material, the writing of this book is not always done in a traditional format. Chapters and sections are numbered in a traditional way. Each section is further divided into subsections, for example, the 3rd subsection of the 2nd section of Chapter 1 is denoted as "1.2.3." Textboxes in a subsection are ordered with numbers enclosed by a pair of square brackets (e.g., [4]). We may refer to such a textbox as "1.2.3[4]." When referring to a textbox in the same subsection, we drop the subsection identifier; for the foregoing example, we simply write "[4]." Equations are numbered in a similar way, except that the equation number is enclosed by a pair of round brackets (parentheses) rather than square brackets. For example, "1.2.3(1)" refers to the 1st equation in the Subsection 1.2.3. Numbering notations are summarized as follows (some of these notations are reiterated in 1.1.1[6] (page 10), 1.1.2[9] (page 12), and 2.1.2[12] (page 58)):

[1], [2], ...	A number enclosed by brackets is used to identify a textbox.
(1), (2), ...	A number enclosed by round brackets is used to identify an equation.
Reference ^[Ref 1]	Superscripts are used for references.
Mechanical	Boldface is used to highlight Workbench keywords.
Round-cornered textboxes	A round-cornered textbox indicates that mouse or keyboard actions are needed.
Sharp-cornered textboxes	A sharp-cornered textbox is for commentary; no mouse or keyboard needed.
→, ←, ↓, ↑, ↘, ↙, ↗, ↖	An arrow is used to point to the location of the next textbox.
↵	This symbol is used to indicate that the next textbox is on the next page.
#	This symbol is used to indicate that it is the last textbox of a section.

Acknowledgements

I feel thankful to students in my classroom listening to my lectures. It is my students, past and present, that motivated me to give birth to this book.

Many of the cases presented in this book are from my students' final-term projects. Some are industry cases while others are thesis-related research topics. Without these real-world cases, the book would never be so useful. The following is a list of students who contributed to the cases in this book.

"Pneumatic Finger" (1.1 and 9.1) is contributed by Che-Min Lin and Chen-Hsien Fan, ME, NCKU.

"Microgripper" (2.6 and 13.3) is contributed by P.W. Shih, ME, NCKU.

"Cover of Pressure Cylinder" (4.2 and 9.2) is contributed by M. H. Tsai, ME, NCKU.

"Lifting Fork" (4.3 and 12.2) is contributed by K.Y. Lee, ES, NCKU.

"LCD Display Support" (4.5 and 5.4) is contributed by Y.W. Lee, ES, NCKU.

"Bellows Tube" (6.1) is contributed by W. Z. Liu, ME, NCKU.

"Flexible Gripper" (7.1 and 8.1) is contributed by Shang-Yun Hsu, ME, NCKU.

"3D Truss" (7.2) is contributed by T. C. Hung, ME, NCKU.

"Snap Lock" (13.4) is contributed by C. N. Chen, ME, NCKU.

Many of the original ideas of these projects came from the academic advisors of the above students. I also owe them a debt of thanks. Specifically, the project "Pneumatic Finger" is a work led by Prof. Chao-Chieh Lan of the Department of ME, NCKU. The project "Microgripper" originates from a work led by Prof. Ren-Jung Chang of the Department of ME, NCKU.

Thanks to Dr. Shen-Yeh Chen, the CEO of FEA-Opt Technology, for letting me use his article "An Unofficial History of ANSYS," as an appendix at the end of Chapter 1.

S.Y. Kan, the chief structural engineer of the Taipei 101, has reviewed some of the text and corrected some mistakes.

Much of the information about the ANSYS Workbench was obtained from the training tutorials prepared by ANSYS Inc. I didn't specifically cite them in the text, but I appreciate these well-compiled training tutorials.

Thanks to Mrs. Lilly Lin, the CEO, and Mr. Nerow Yang, the general manager, of Taiwan Auto Design, Co., the partner of ANSYS, Inc. in Taiwan. The couple, my long-term friends, provided much substantial support.

Thanks to Professor Sheng-Jye Hwang, of the ME Department, NCKU, and Professor Durn-Yuan Huang, of Chung Hwa University of Medical Technology. They are my long-term research partners. Together, we have accomplished many projects, and, in carrying out these projects, I've learned much from them.

Lastly, thanks to my family, my wife and my son and the pets, for their patience and sharing the excitement with me during the writing of this book.

Huei-Huang Lee

Associate Professor
Department of Engineering Science
National Cheng Kung University, Tainan, Taiwan
e-mail: hhlee@mail.ncku.edu.tw
webpage: myweb.ncku.edu.tw/~hhlee

Chapter I

Introduction

Purpose of This Chapter

ANSYS is a software implementation of finite element simulations on many types of problems, including structural, mechanical, fluid, electromagnetic, etc. This book discusses only structural and mechanical simulations. This chapter shows the procedure of a typical structural simulation using Workbench (Section 1.1), explains the organization of this book (Section 1.1), reviews solid mechanics (Sections 1.2 and 1.4), and gives a brief introduction of finite element methods (Section 1.3).

About Each Section

The procedure of structural simulations and the organization of this book is illustrated in Section 1.1 by using a case study. Section 1.1 also serves as a preamble for the topics in Sections 2, 3, and 4.

Section 1.2 introduces quantities such as displacements, stresses, and strains, which are used throughout the book. With these quantities, the equations that govern the behavior of a structural system are presented.

ANSYS solves these governing equations by finite element methods. Section 1.3 introduces the basic ideas and the procedure of the finite element methods. The introduction is conceptual rather than theoretical or mathematical. These concepts should be adequate for the purpose of understanding the topics in the later chapters.

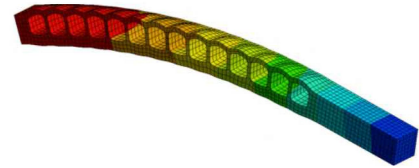
One goal of structural simulations is to predict whether or not a system would fail. We usually compare the calculated stresses with certain critical values. If the calculated stresses are larger than the critical values, then the system is said to fail. Section 1.4 discusses the theories of failure criteria.

This is the Only Chapter without Hands-On Exercises

All chapters of this book use the learning-by-doing approach, except this chapter. There are no hands-on exercises in this chapter. The main reason is that an overall picture is usually helpful before any hands-on exercises. A secondary reason is that, in the first week of a semester, students may not be ready to access the software facility.

Section 1.1

Case Study: Pneumatically Actuated PDMS Fingers^[Ref 1]



This section demonstrates a typical ANSYS Simulation procedure and introduces the organization of the book. There are no step-by-step exercises in this section; you don't have to sit in front of a computer when reading this section. In Section 9.1, you will be guided to conduct this simulation.

1.1.1 Problem Description

About the Pneumatic Fingers

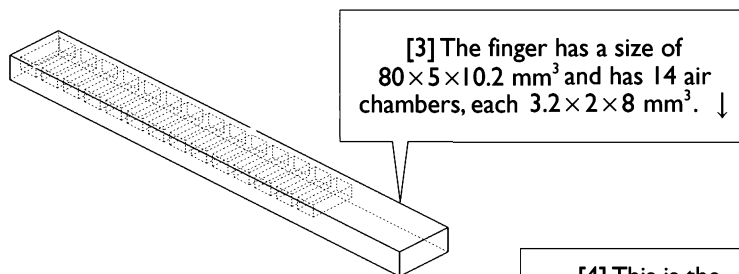
[1] The pneumatic fingers (see [2]) are designed as part of a surgical parallel robot system remotely controlled by a surgeon through the internet^[Ref 2].

The fingers are made of a PDMS-based (polydimethylsiloxane) elastomer material. The geometry of a typical finger is shown in [3], in which 14 air chambers are cut from the material.

The air chambers locate close to the upper surface, so when the air pressure is applied, the finger bends downward [4-5]. →

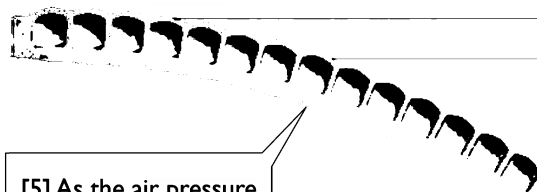


[2] A robotic hand has five fingers, remotely controlled by a surgeon through the internet. ←



[3] The finger has a size of $80 \times 5 \times 10.2 \text{ mm}^3$ and has 14 air chambers, each $3.2 \times 2 \times 8 \text{ mm}^3$. ↓

[4] This is the undeformed shape. ✓



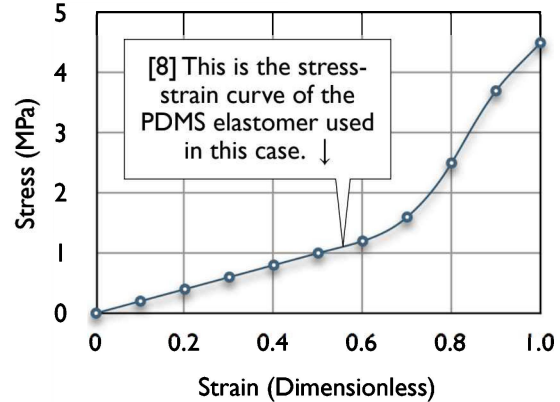
[5] As the air pressure is applied, the finger bends downward. →

About Textboxes

[6] In this book, textboxes within a subsection (e.g., 1.1.1) are ordered with numbers enclosed by square brackets (e.g., [1]). When you read, please follow the order of the textboxes. An arrow is used at the end of a textbox to locate the next textbox (e.g., →, ←, ↓, ↗, and → in [1-5]). The symbol ↵ is used to indicate that the next textbox is on the next page (e.g., [6]). The symbol # is used to indicate that it is the last textbox of a subsection (e.g., [9], next page). ↵

About the PDMS Elastomer

[7] The mechanical properties of a PDMS elastomer are functions of the ingredients. The chart in [8] shows the stress-strain curve of the PDMS elastomer used in this case^[Ref 3]. The curve exhibits a linear stress-strain relationship up to a strain of about 0.6. Within this range, the Young's modulus (the slope of the curve) is estimated to be 2.0 MPa (1.2 MPa divided by 0.6). Besides, the Poisson's ratio is 0.48, from another test. →



Purpose of the Simulation

[9] The purpose of this simulation is to assess the *efficiency* of the design, defined as the vertical deflection under an air pressure of 0.18 MPa. We also want to plot a deflection-versus-pressure chart (1.1.8[6], page 19; also see 1.1.12[3-4], page 23). #

1.1.2 Workbench GUI

[1] In a Windows system, you can launch Workbench from the **Start** menu or double-click a Workbench icon (if there is one) on the desktop [2]. A **Workbench GUI** (graphic user interface) then shows up [3]. The **Workbench GUI** is a gateway to Workbench applications. The Workbench applications that will be used in this book are **Project Schematic**, **Engineering Data**, **DesignModeler**, **Mechanical**, and **Design Exploration**. ✓

[3] **Workbench GUI.** shows up. →

[2] Launch Workbench. ↑

[4] **Toolbox.** →

[5] **Project Schematic.** ↓

[6] Double-click to create a **Static Structural** analysis system. ↗

[7] A **Static Structural** analysis system is created in **Project Schematic**. The six cells in the analysis system indicate that six steps are needed to perform a static structural analysis. ↶

[8] On the left of the **Workbench GUI** is a **Toolbox** ([4], last page), and on the right is a **Project Schematic** window [5]. **Toolbox** lists available **Analysis Systems**, which may be different in your computer from here, depending on your installation. In this book, "analysis" and "simulation" are often interchangeable, for example, "static structural analysis" is synonymous to "static structural simulation." Here, we want to perform a **Static Structural** analysis.

Double-clicking **Static Structural** [6] in **Toolbox** creates a **Static Structural** analysis system [7] in the **Project Schematic** window. The six cells in the analysis system indicate that six steps are needed to perform a static structural analysis: (a) prepare engineering data, (b) create a geometric model, (c) divide the geometric model into a finite element mesh, which is also called a finite element model, (d) set up loads and supports, (e) solve the finite element model, and (f) view the results.

Double-clicking each cell will bring up a relevant application to process that step. ↓

[9] In this book, to facilitate the readability, a Workbench keyword is usually boldfaced, e.g., **Static Structural** in [8]. In cases where boldface does not help the readability, then the boldface may not be necessary, e.g., Workbench in [2]. #

1.1.3 Prepare Engineering Data

[1] Double-clicking **Engineering Data** cell [2] brings up **Engineering Data** [3]. Here, we want to specify material properties for the PDMS elastomer. The material is modeled as a linear **Isotropic Elasticity** material, for which we need to input a Young's modulus (2.0 MPa) and a Poisson's ratio (0.48) [4-6]. At completion, close the **Engineering Data** application [7] and return to **Project Schematic**. ✓

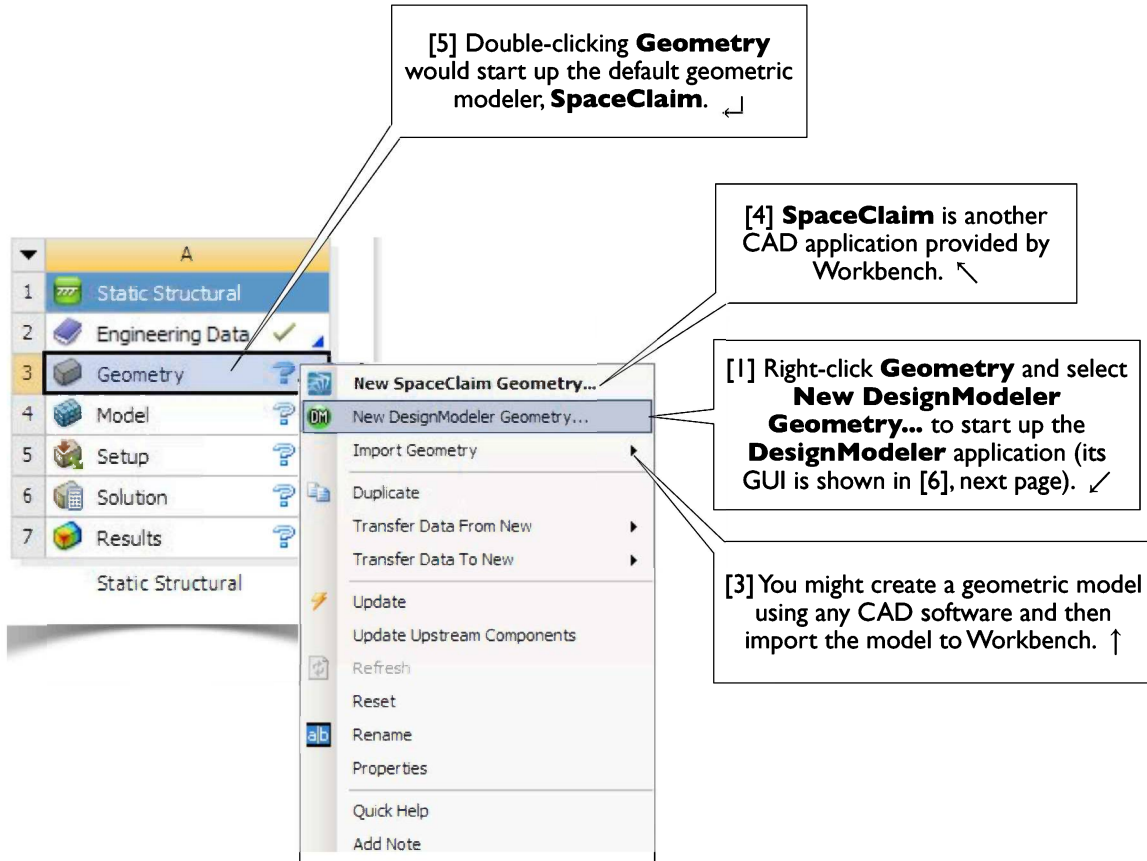
The screenshot shows the ANSYS Workbench interface with several callouts explaining the steps:

- [2] Double-click to start up the **Engineering Data** application. →** Points to the 'Engineering Data' cell in the Project Schematic.
- [5] Double-click **Isotropic Elasticity** to include this material model. ↘** Points to the 'Isotropic Elasticity' option in the Toolbox.
- [3] The **Engineering Data** application shows up. ↓** Points to the 'Engineering Data Sources' window.
- [4] Type a name to add a new material to **Engineering Data**. ←** Points to the 'PDMS' entry in the 'Outline of Schematic A2: Engineering Data' table.
- [6] Type the values for the Young's modulus and Poisson's ratio. ↘** Points to the 'Properties of Outline Row 4: PDMS' table, specifically the 'Young's Modulus' and 'Poisson's Ratio' fields.
- [7] Click to close the application and return to **Project Schematic**. #** Points to the 'Close' button in the 'Engineering Data Sources' window.

The 'Properties of Outline Row 4: PDMS' table is shown below:

Property	Value	Units
Material Field Variables		
Isotropic Elasticity		
Derive from	Young's Modulus and Poisson's Ratio	
Young's Modulus	2E+06	Pa
Poisson's Ratio	0.48	
Bulk Modulus	1.6667E+07	Pa
Shear Modulus	6.7568E+05	Pa

1.1.4 Create Geometric Model



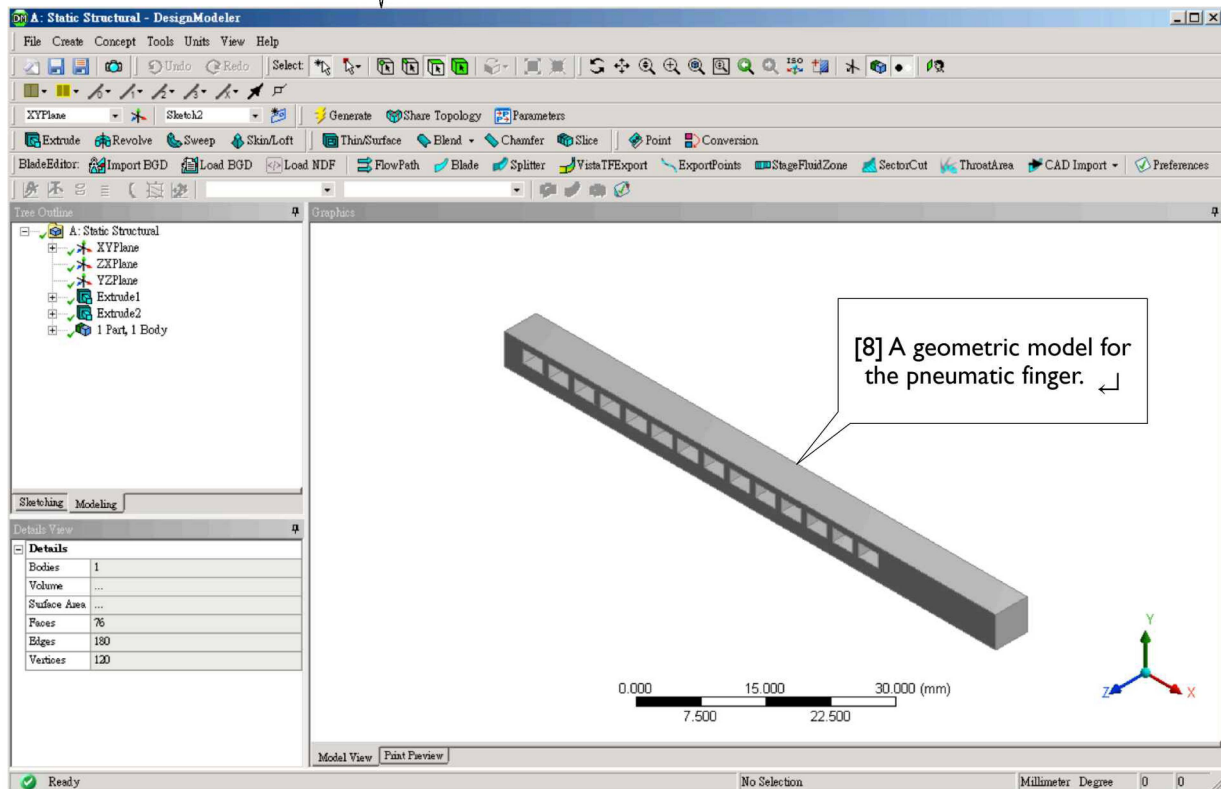
Ways of Creating Geometric Models

[2] You might create a geometric model using any CAD software (e.g., SOLIDWORKS, PTC Creo, Autodesk Inventor) and then import the model to Workbench [3]. However, I suggest that you create a geometric model using one of the two CAD applications provided by Workbench, since they are specifically designed for use in ANSYS Workbench simulations.

The two CAD applications in Workbench are **DesignModeler** [1] and **SpaceClaim** [4]. Until ANSYS 15, DesignModeler is the only built-in geometric modeler in Workbench. For simple and small models, DesignModeler serves well enough; but for complicated and large models, the engineers often resort to a CAD software such as SOLIDWORKS, PTC Creo, Autodesk Inventor, etc., and then import the model to Workbench [3]. In ANSYS 16 and 17, **SpaceClaim** was included in Workbench as a second modeler, and **DesignModeler** remained as the default modeler. Starting from ANSYS 18, **SpaceClaim** becomes the default modeler [5], and **DesignModeler** serves as an alternative modeler.

In this book, since all the geometric models are simple and small, we will use DesignModeler to create the geometric models. Remember, the focus of this book is the finite element simulations, not the geometric modeling. ↑

[6] DesignModeler. ↓



[7] The functions of DesignModeler are similar to any other CAD software except, as we mentioned, that DesignModeler is specifically designed to create geometric models for use in ANSYS Workbench simulations. Due to the symmetry, we will create only one half of the pneumatic finger, as shown in [8].

Creating the geometric model in our case is relatively simple. It can be viewed as a two-step task, and each step consists of a 2D **sketching** and a 3D **extrusion**. The procedure is summarized as follows.

Step 1. Create an 80 mm x 5 mm x 5.1 mm solid.

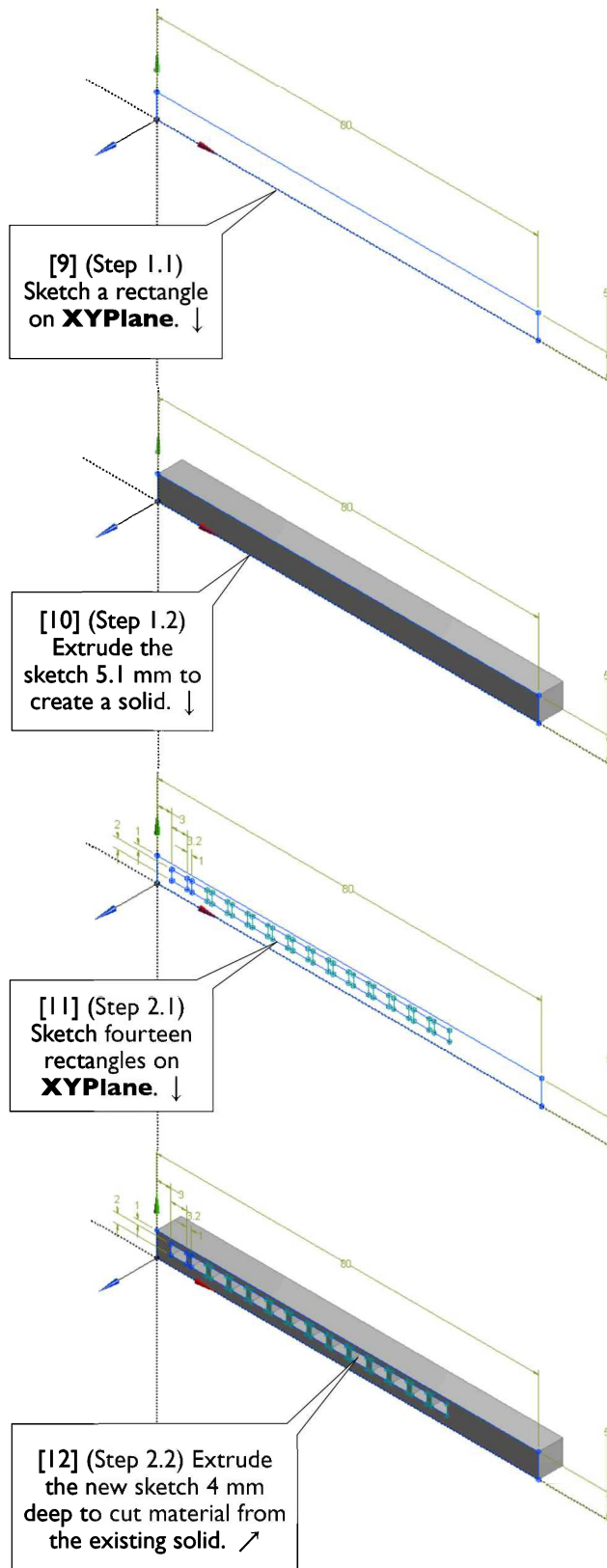
Step 1.1. Sketch an 80 mm x 5 mm rectangle ([9], next page).

Step 1.2. Extrude the sketch 5.1 mm to create a solid [10].

Step 2. Create fourteen 3.2 mm x 2 mm x 4 mm air chambers.

Step 2.1. Sketch fourteen 3.2 mm x 2 mm rectangles [11].

Step 2.2. Extrude the new sketch 4 mm deep to cut material from the existing solid [12]. ↑



2D and 3D Simulations

[13] Workbench supports 2D and 3D simulations. For 3D simulations, Workbench supports three types of geometric bodies: *solid bodies* (which have volumes), *surface bodies* (which do not have volumes, but have surface areas), and *line bodies* (which do not have volumes or surface areas, but have length). Thin shell structures are often modeled as surface bodies (Chapter 6); Beam or frame structures are often modeled as line bodies (Chapter 7). Solid, surface, and line bodies can be mixed up in a 3D model. For 2D simulations, Workbench supports solid bodies only (Chapter 3). A 2D model must be created entirely on **XYPlane**. ↓

More on Geometric Modeling

[14] Creating a geometric model is usually more involved than that example; nonetheless, often, it still can be viewed as a series of two-step operations demonstrated in this case: drawing a sketch and then using the sketch to create a 3D body by one of the modeling tools, such as extrusion, revolution, sweeping, and skin/lofting (4.4.8[1], page 201).

Geometric modeling is the first step toward successful finite element simulations. Chapters 2 and 4 demonstrate some geometric modeling techniques using DesignModeler. Chapter 2 focuses on sketching methods. Some of the sketches created in Chapter 2 are reused in Chapter 3 to demonstrate the creation of 2D solid models. Simulations of these 2D solid models are then performed in Chapter 3.

Chapters 4-7 discuss 3D geometric modeling and linear static simulations (except 7.1, which involves geometric nonlinearities). Chapter 4 demonstrates the creation of 3D solid models and Chapter 5 demonstrates 3D simulations using the solid models created in Chapter 4. Chapter 6 demonstrates 3D surface modeling and simulations, and Chapter 7 demonstrates 3D line modeling and simulations.

In the real-world, there are no such things as surface/line geometries, therefore surface/line models (as well as 2D solid models) are called **conceptual models**. #

1.1.5 Divide the Geometric Model into Elements

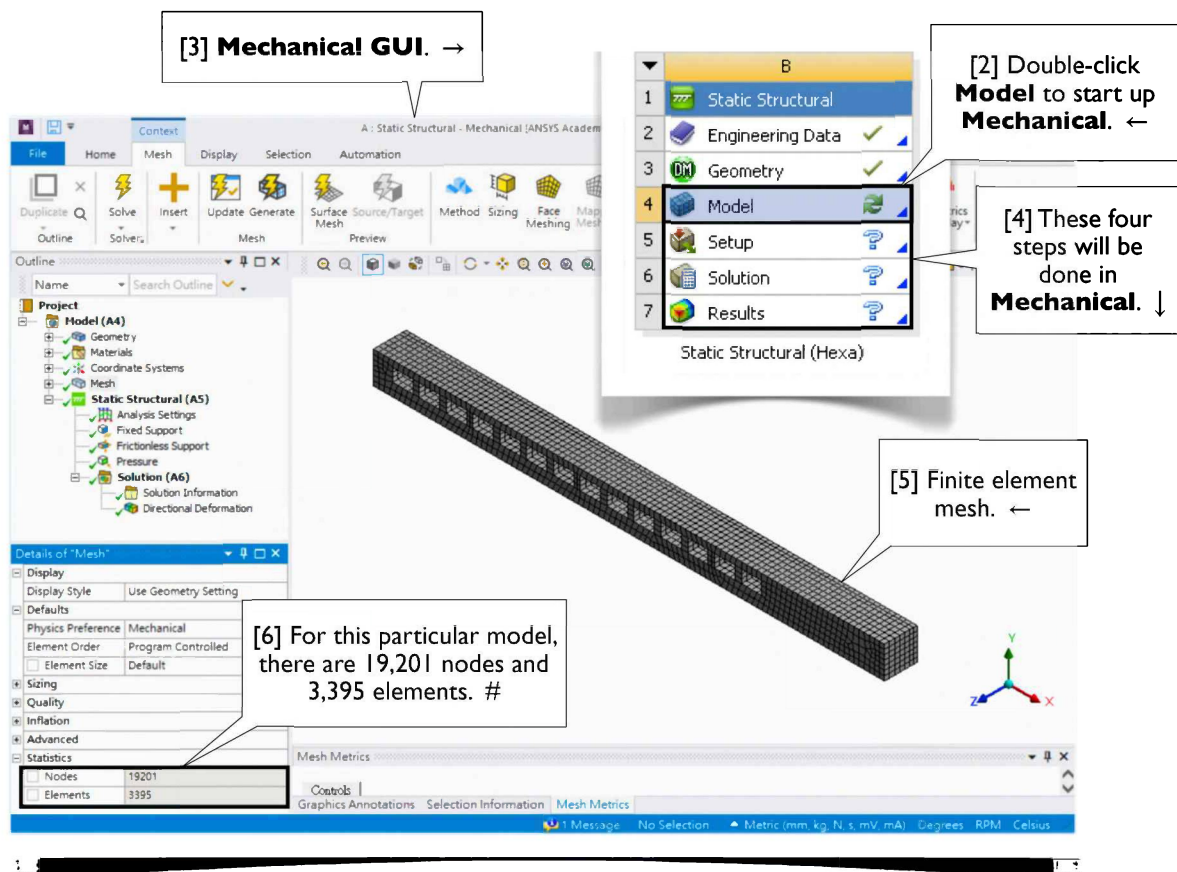
[1] The procedure used by Workbench to solve a problem is called a **finite element method**, which can be viewed as two major steps: (a) establishing governing equations and (b) solving the governing equations. The geometric body of a real-world problem is often too complicated to write down the governing equations directly. A basic idea of the finite element method is to divide the body into many geometrically simpler shapes called *elements*. The elements are connected by *nodes*. The governing equations for each element then can be easily established, and the system of equations for all elements can be solved simultaneously. We will discuss this idea further in Section 1.3.

Since the elements have finite size (in contrast to the infinitesimal sizes of elements in Calculus), they are called **finite elements**. The collection of the elements is called a **finite element mesh**; in Workbench, a finite element mesh is also called a **finite element model**. In this book, we define the *finite element model* as a *finite element mesh plus its environment conditions* (loads and supports). The environment conditions are introduced in 1.1.6, next page.

Double-clicking the **Model** cell [2] in **Static Structural** brings up a **Mechanical GUI** [3]. The rest of the simulation will be done in **Mechanical**; that is, the functions of **Mechanical** include meshing, set up of loads and supports, solution, and viewing results [4].

Workbench can perform meshing task under your control [5]. In **Details of Mesh, Statistics** displays the number of nodes and elements [6].

The quality of a mesh cannot be overemphasized. Although it is possible to let ANSYS Workbench perform the meshing automatically, its quality is not guaranteed. Achieving a high quality mesh is not trivial; it needs much background knowledge and experience. Chapter 9 demonstrates many meshing techniques. ↓



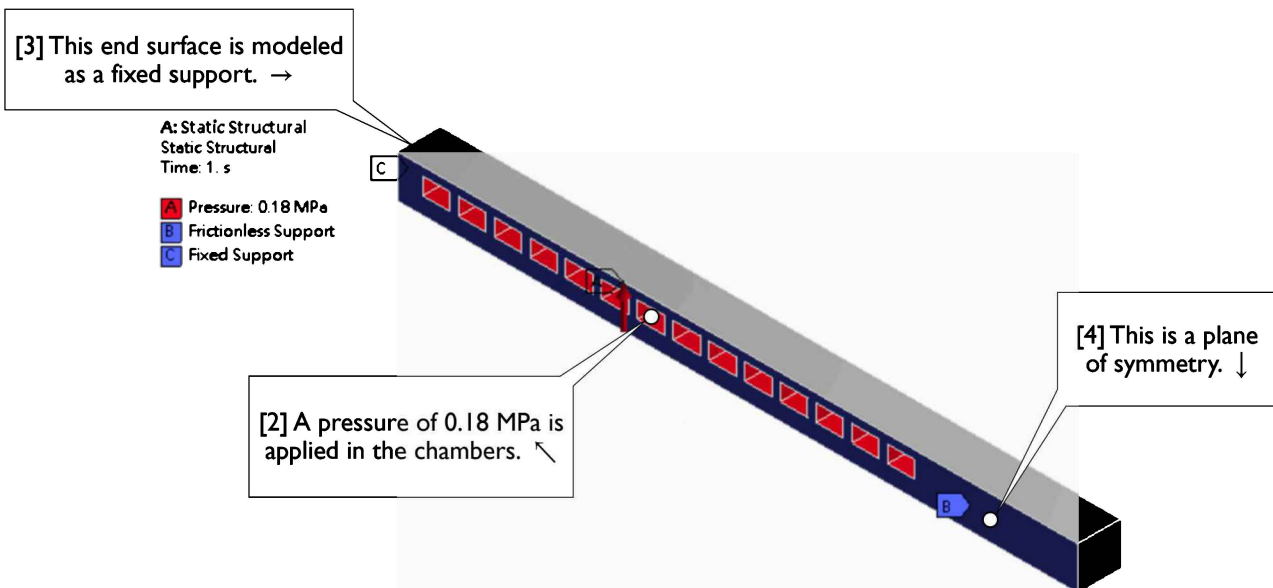
1.1.6 Set up Loads and Supports

[1] In the real-world, a body is a part of the world, interacting with other parts of the world. When we take a body apart for simulation, we are cutting it away from the rest of the world. The cutting surfaces are the boundary surfaces of the body. In theory, the choice of the boundary surfaces is arbitrary; however, we need to specify the *boundary conditions* on ALL of the boundary surfaces. In Workbench, conditions applying on the finite element mesh are called the *environment conditions*, including the boundary conditions and the conditions that are not specified on the boundaries. The temperature change INSIDE a body is an example of environment conditions that are not specified on the boundaries; another example is the gravitational forces.

In our case, things surrounding the half model of the pneumatic finger are pressurized air in the chambers [2], the material connecting the root of the finger [3], the material on the other side of the model [4], and the atmosphere air around the rest of the boundary surfaces.

Modeling the pressurized air in the chambers is straightforward: specify a pressure of 0.18 MPa for all the surfaces of the chambers [2]. The root of the finger is modeled as fixed support [3]. The plane of symmetry is modeled as frictionless support [4]. Note that a plane of symmetry is equivalent to a surface of frictionless support. Finally, assuming the atmospherical air has little effect on the model, we simply neglect it and model all the boundary surfaces surrounded by atmospherical air as free boundaries, i.e., boundaries with no forces acting on them.

As mentioned (1.1.5[1], last page), a finite element model is defined as a finite element mesh plus its environment conditions. We will stick to this definition throughout the book. Make sure that you can distinguish these three terms: *geometric model*, *finite element mesh*, and *finite element model*. ↓



More on Environment Conditions

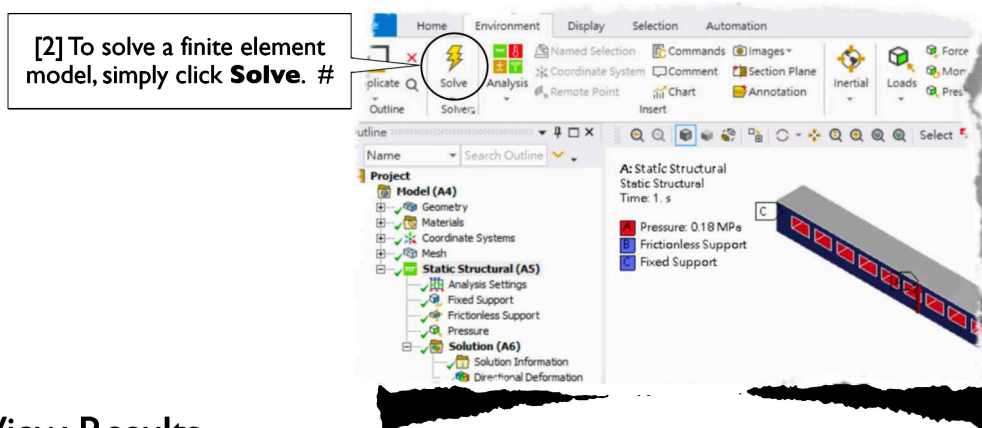
[5] Modeling environment conditions is sometimes not so easy as in this case. The challenge comes from the need of domain knowledge. It is not possible to perform a structural analysis if an engineer doesn't have enough domain knowledge of structural mechanics.

We will start to introduce environment conditions in Chapter 3. From then on, each chapter will involve some demonstrations of environment conditions. #

1.1.7 Solve Finite Element Model

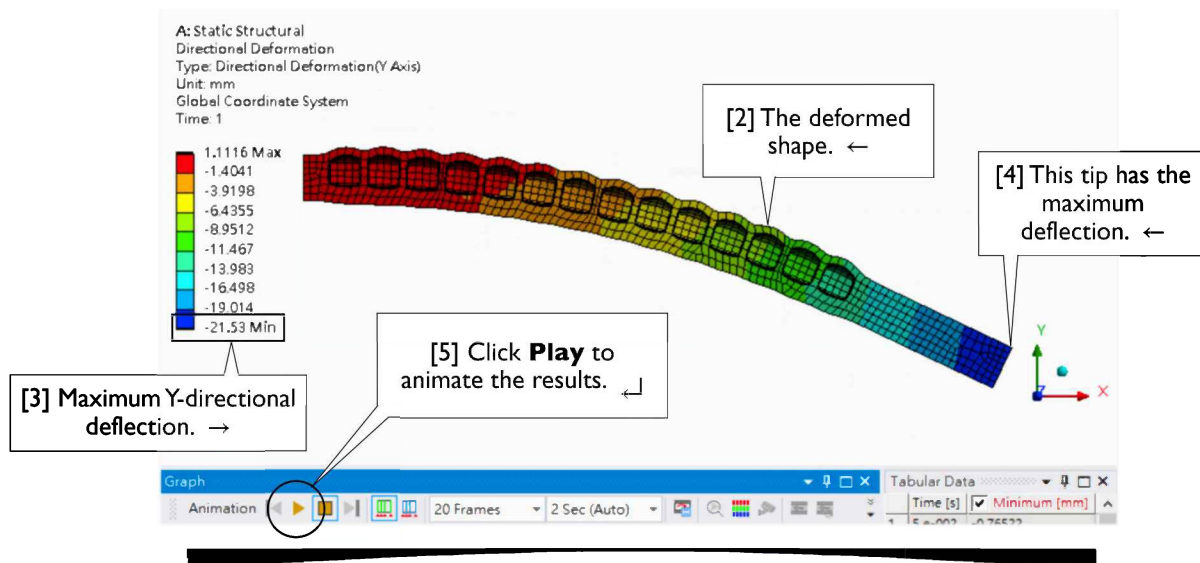
[1] To solve a finite element model, simply click **Solve** in **Mechanical GUI** [2]. The time to complete a simulation depends on its problem size (number of nodes and number of elements), number of time steps, and nonlinearities.

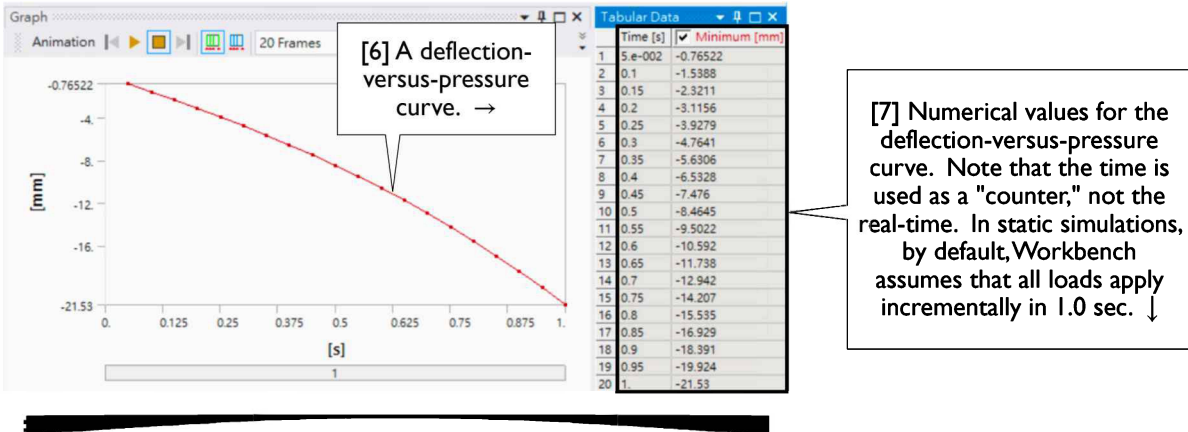
As mentioned (1.1.5[1], page 16), the solution procedure can be viewed as two major steps: establishing governing equations and solving the governing equations. Section 1.2 overviews structural mechanics and summarizes the governing equations. How does ANSYS Workbench establish and solve these governing equations? The answer is *finite element methods*. Section 1.3 summarizes finite element methods, quickly equipping the students with enough concepts of the methods so they can proceed to learning the materials in this book. ↓



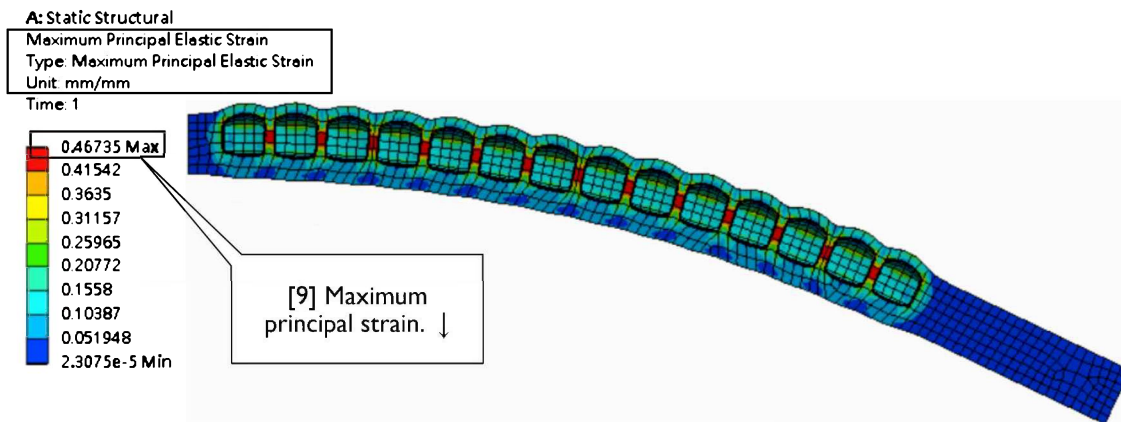
1.1.8 View Results

[1] After solving a problem, numerical results are stored in the databases, available for your request. In our case, we are concerned about the vertical deflection [2-4]. The deformation can be animated [5]. A deflection-versus-pressure chart is useful ([6-7], next page), in which the deflection is measured at the tip of the finger [4]. ↓





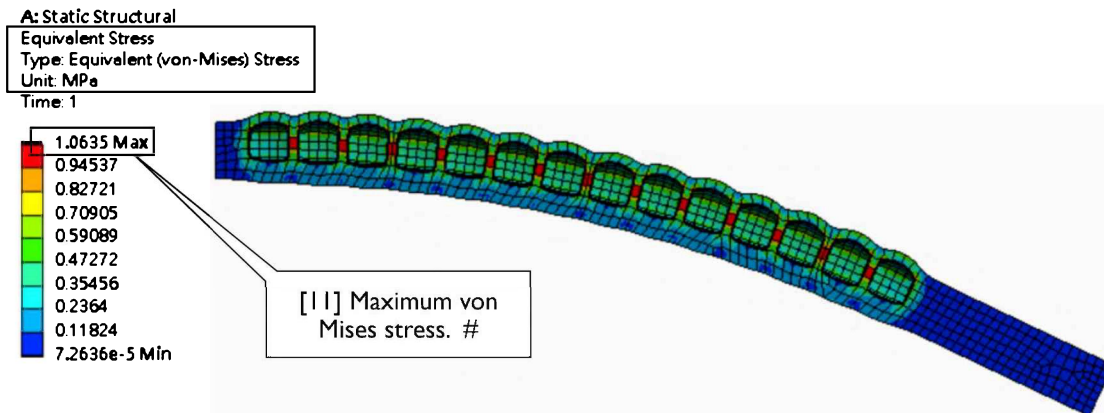
[8] In this case, we assume the stress-strain relationship of the PDMS material is linear (1.1.3[5], page 12). This implies that the strains are less than 0.6 (1.1.1[7-8], page 11). This assumption has to be verified at this point. The results show that the maximum strain is about 0.47 [9], well below 0.6. Note that we also assumed the compressive behavior is the same as tensile behavior, but this is usually not true for an elastomer under such a large deformation. A more accurate material model, a hyperelasticity model, for the elastomer will be introduced in Chapter 14. ↓



[10] Lastly, the stress is reviewed. It shows that the maximum von Mises stress, defined in Eq. 1.4.5(15) (page 46), is 1.06 MPa ([11], next page). The test data (1.1.1[8], page 11) show that the material can withstand up to 4.5 MPa without failure. The design is safe as far as the stress is concerned.

Failure Criteria

The main purpose of checking the stresses is to make sure the material doesn't fail under specified load. What is von Mises stress ([11], next page)? The test data (1.1.1[8], page 11) are produced according to a uniaxial tensile test, but the stress state in the pneumatic finger, as in any other real-world situations, is 3D by nature. How can we compare a 3D stress state with a uniaxial one, and judge the failure of the material? Section 1.4 will discuss failure criteria of materials. ↵



1.1.9 Buckling and Stress-Stiffening

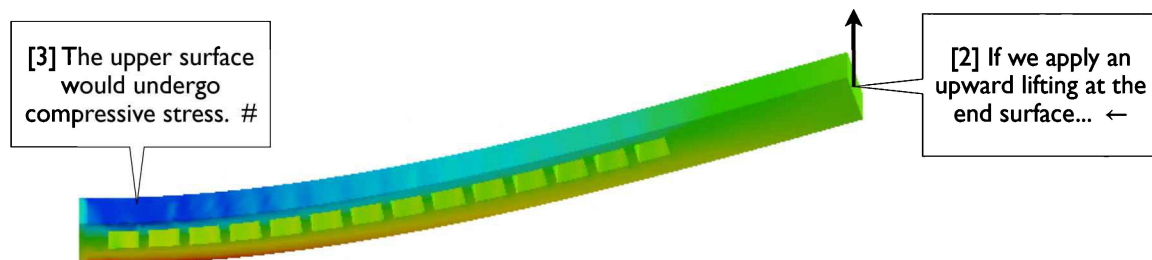
[1] The more tension in a guitar string, the more force you need to deflect the string laterally. In technical words, the string's lateral stiffness increases as the longitudinal tensile stress increases; i.e., the longitudinal tensile stress causes the string to be stiffer in lateral direction. By lateral direction, we mean the direction orthogonal to the longitudinal direction. The increase of lateral stiffness can also be justified by the fact that the string's vibrating frequency (pitch) increases with the increase of its tension, since stiffer strings have higher frequencies. This effect is called the *stress-stiffening effect: a structure's lateral stiffness increases with the increase of its longitudinal tensile stress*.

Is the opposite also true? That is, does the lateral stiffness of a structure decrease with the increase of its axial compressive stress? For example, does a column's lateral stiffness decrease when subject to an axial compressive force? The answer is YES. An even more dramatic phenomenon is that, as the compression is increasing and the lateral stiffness is decreasing, the lateral stiffness will eventually reach zero and the structure is said to be in an unstable state, in which a tiny lateral force would deflect the structure infinitely. This phenomenon is called *buckling*.

The buckling must be considered in a compressive structural component, particularly when its lateral dimension is much smaller than the longitudinal dimension; for example, slender columns subject to axial loads, thin-walled pipes subject to a circumferential twist.

Back to the pneumatic finger. Instead of air pressure applied on the chambers' surfaces, we now apply an upward lifting at the finger tip [2]. The upper surface, which is essentially a layer of thin PDMS film, would undergo compressive stress [3]. Our concern then is to know the magnitude of the lifting force that will cause the thin film buckle.

Simulations of buckling and stress-stiffening will be covered in Chapter 10. ↓

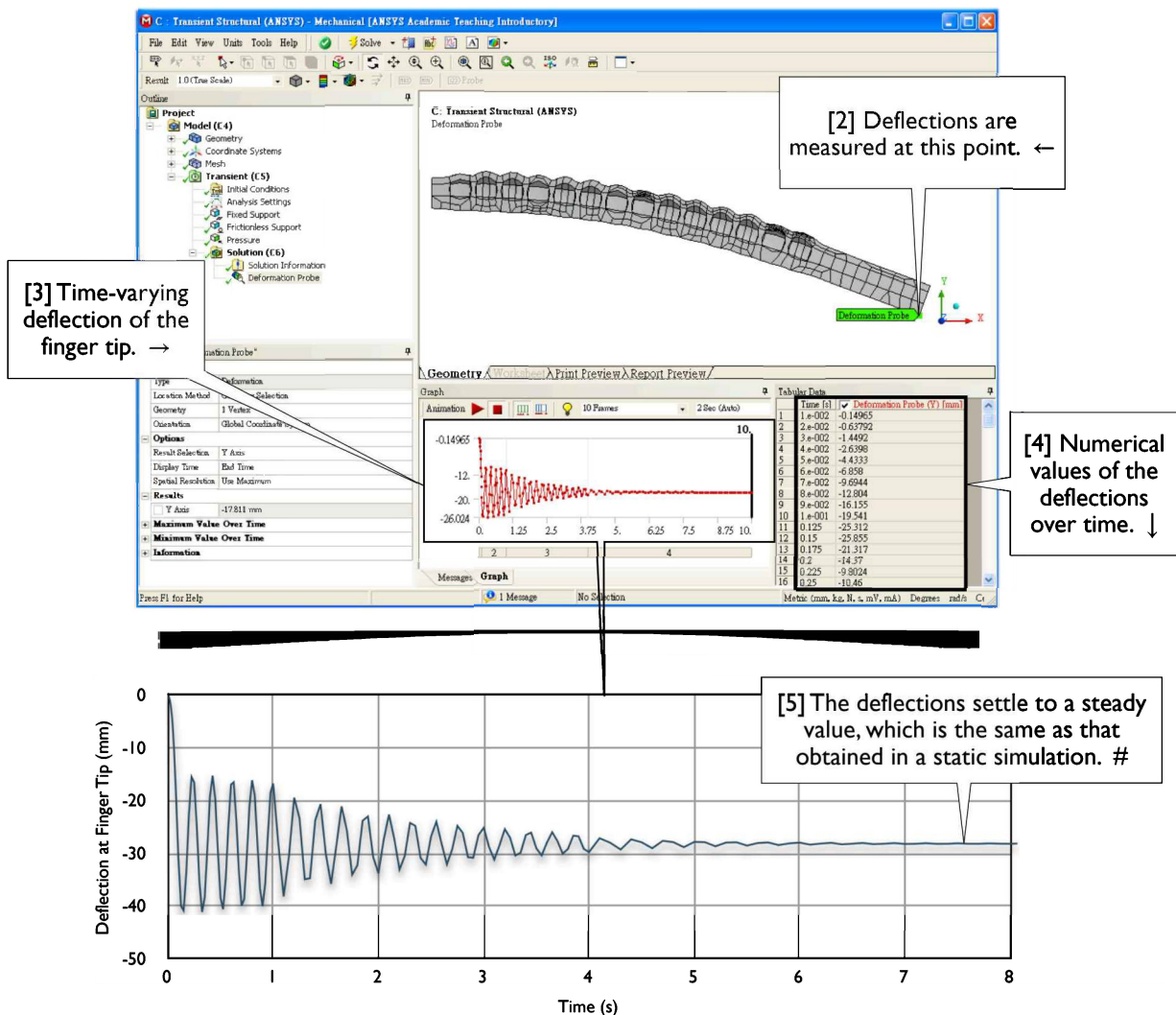


1.1.10 Dynamic Simulations

[1] If the load (air pressure) is applied to the chambers very fast, the deformation would also occur very fast. When a body moves or deforms very fast, two effects must be taken into account: *inertia effect* and *damping effect*. Combination of these two effects is called *dynamic effects*. When the dynamic effects are considered in a simulation, it is called a *dynamic simulation*.

Imagine that the pressure in the chambers is increased from zero to 0.18 MPa in just 0.1 seconds. The pressure is applied so fast that the deformation must also be very fast and dynamic effects must play an important role in the structure's behavior. The figure and the chart below show the time-varying deflections of the finger tip under such a loading condition [2-5]. The curve [5] shows that the deflections in a dynamic simulation can be much larger than those obtained in a static simulation. Furthermore, the vibration lasts for several seconds. This, as a surgical application, is not desirable.

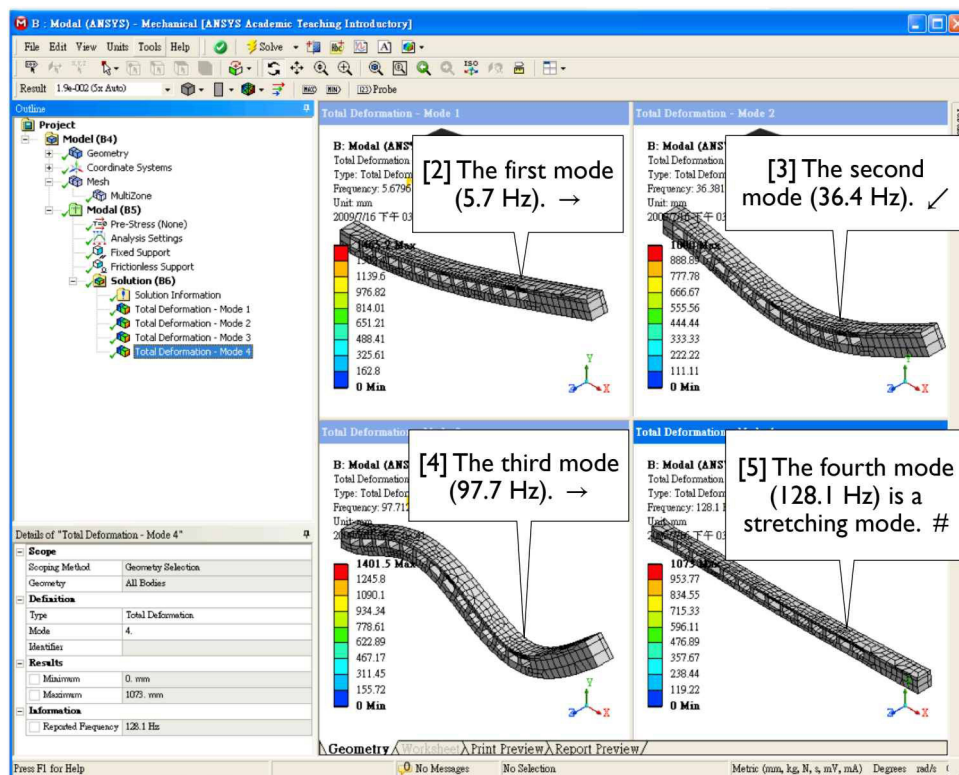
The foregoing simulation, a structure subject to dynamic loads, is called a *transient structural simulation*. Chapters 12 and 15 will cover transient structural simulations: Chapter 12 discusses *implicit methods* while Chapter 15 introduces *explicit methods*. →



I.1.11 Modal Analysis

[1] A special case of dynamic simulations is the simulation of *free vibrations*, the vibrations of a structure without any external loading (but prestress is allowed). Consider that you deflect a structure and then release, causing the structure to vibrate without external forces. We want to know the behavior of this free vibration. The simulation is called a *modal analysis*. The results of modal analysis include the *natural frequencies* and the *vibration modes* of the structure. The figure below [2-5] shows the four lowest natural frequencies and the corresponding vibration modes.

A modal analysis is much less expensive (in terms of engineer's work hours and computing time) than a transient analysis and is often performed before a transient analysis to obtain preliminary dynamic characteristics of a structure. Chapter II will discuss modal analysis and its applications. ↓



I.1.12 Structural Nonlinearities

[1] When the responses (deflection, stress, strain, etc.) of a structure are linearly proportional to the loads, the structure is called a *linear structure* and the simulation is called a *linear simulation*. Otherwise the structure is called a *nonlinear structure* and the simulation is called a *nonlinear simulation*.

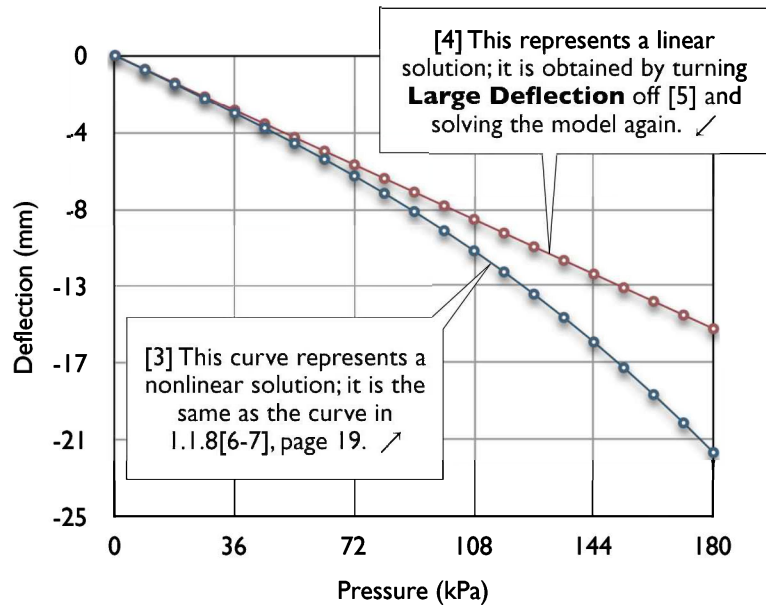
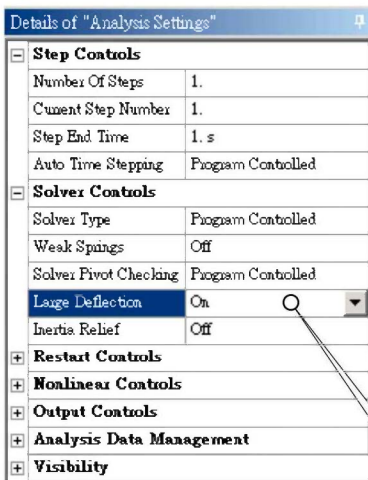
Structural nonlinearities commonly come from three sources: (a) Due to large deformation; this is called *geometry nonlinearity*. (b) Due to the topological change of the structure; this is called *topology nonlinearity*. A common case of topology change is the change of contact status, and is called *contact nonlinearity*. (c) Due to the nonlinear stress-strain relationship of the material; this is called *material nonlinearity*. ↵

[2] In this case, the stress-strain relationship of the PDMS is almost linear within the range of operational air pressures, and there is no contact between any parts; therefore, there is no material nonlinearity or contact nonlinearity. It, however, has a deflection so large that it exhibits a certain degree of geometry nonlinearity.

The curve in [3] is reproduced from the curve in 1.1.8[6-7], page 19. It shows a nonlinear relationship between the deflection and the pressure. For comparison, we also include a linear solution [4]. The linear solution is obtained by turning **Large Deflection** off [5] and solving the model again.

A comparison between the nonlinear solution [3] and the linear solution [4] concludes that, in this case, the error would be significant if geometry nonlinearity were not taken into account.

Solving a nonlinear problem is often challenging and sometimes frustrating. Real-world problems often involve nonlinearities to some degree. We will experience nonlinear simulations as early as in Section 3.2, without detailed discussions of nonlinear solution controls. Chapters 13 and 14 are dedicated to the discussion of nonlinear simulations; Chapter 13 discusses general nonlinear solution methods and covers geometry nonlinearity and contact nonlinearity, while Chapter 14 discusses material nonlinearity. ↓



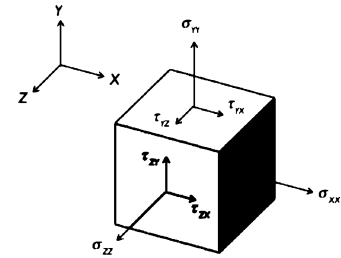
[5] **Large Deflection** option can be used to turn on/off geometry nonlinearity. #

References

1. This case study is adapted from a work led by Prof. Chao-Chieh Lan of the Department of Mechanical Engineering, National Cheng Kung University, Taiwan.
2. Jeong, O. K and Konishi, S., "All PDMS Pneumatic Microfinger With Bidirectional Motion and Its Application," *Journal of Microelectromechanical Systems*, Vol. 15, No. 4, August 2006, pp. 896-903.
3. Draheim, J, Kamberger, R., and Wallrabe, U, "Process and material properties of polydimethylsiloxane (PDMS) for Optical MEMS," *Sensors and Actuators A: Physical*, Vol. 151, Issue 2, April 2009, pp. 95-99.

Section 1.2

Structural Mechanics: A Quick Review



This section (a) defines a structural analysis problem, so the students have a clear picture about the input and the output of a structural analysis system; (b) introduces basic terminology that will be used throughout the book, such as displacements, stresses, and strains; (c) and summarizes equations that govern the behavior of the structure, so the students know what equations Workbench is solving.

To simplify the discussion, this review is limited to homogeneous, isotropic, linear static structural analyses. That is, (a) the material is assumed to be homogeneous, isotropic, and linearly elastic. Assumption of linear elasticity implies that Hooke's law is applicable; (b) the deformation is small enough so that we assume a linear relationship between the displacements and the strains; (c) there are no changes of topology; specifically, there are no changes of contact status during the deformation; (d) the deformation is slow enough so that the dynamic effects can be neglected.

The concepts introduced in this section can be generalized to include non-homogeneous, anisotropic, nonlinear, dynamic problems.

1.2.1 Structural Analysis Problems

[1] Engineering analysis (e.g., structural, mechanical, flow, electromagnetic) is to find the *responses* of a *problem domain* subject to *environmental conditions*.

In structural analyses, the problem domain consists of *solid bodies*; the environmental conditions include *loads* and *supports*; the responses can be described by the *displacements*, *stresses*, or *strains*.

For the pneumatic fingers case (Section 1.1), the problem domain is a body made of the PDMS elastomer. There are two support conditions: fixed support at one of the end faces and frictionless support at the face of symmetry. There is one load: the air pressure applied on the faces of the air chambers.

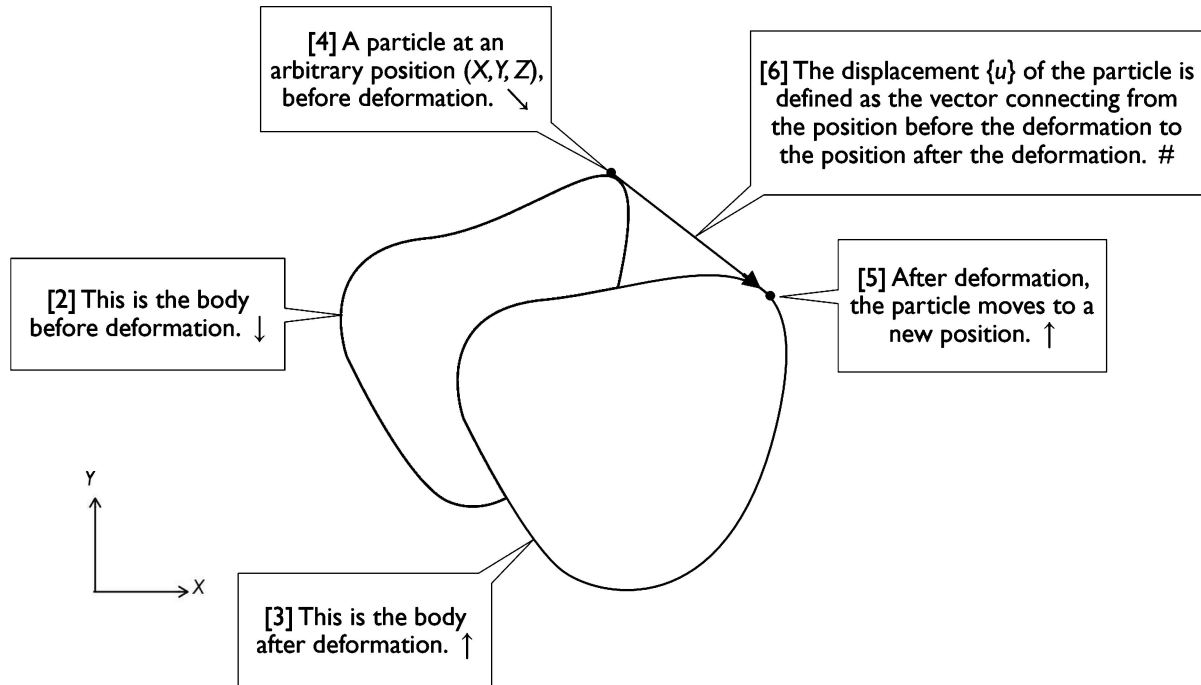
Note that these environmental conditions are applied on boundary faces, so these conditions are also called *boundary conditions*. Environmental conditions may not be applied on boundary surfaces. Common environmental conditions that do not apply on boundary surfaces include temperature changes and inertia forces; these loads distribute over the volumes (rather than boundary faces) of the problem domain. #

1.2.2 Displacement

[1] Deformation of a body can be described by a *displacement field* $\{u\}$ ([2-6], next page). Note that $\{u\}$ is a function of positions and, since it is a vector, we may express the displacement with three components,

$$\{u\} = \begin{Bmatrix} u_x & u_y & u_z \end{Bmatrix} \quad (1)$$

The three components are, of course, functions of positions. The SI unit for displacements is meter (m). ↵



1.2.3 Stress

[1] The displacement is relatively easy to understand, since the displacement can be defined by a vector, and most college students are familiar with the mathematics of vectors. In contrast, the concepts of stress are not so obvious.

Stresses are quantities to describe the intensity of force in a body (either solid or fluid). Its unit is force per unit area (i.e., N/m^2 in SI). It is a position-dependent quantity.

Imagine that your arms are pulled by your friends with two forces of the same magnitude but opposite directions. What are the stresses in your arms? Assuming the magnitude of the forces is P and the cross-sectional area of your arms is A , then you may answer, "the stresses are P/A , everywhere in my arms." This case is simple and the answer is good enough. For a one-dimensional case like this, the stress σ may be defined as $\sigma = P/A$, where P is the applied force and A is the cross sectional area.

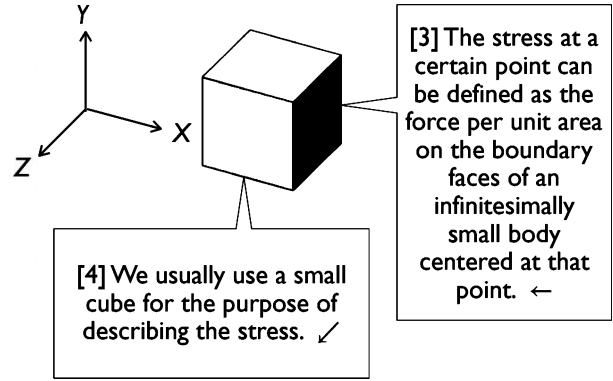
In 3D cases, things are much more complicated. Now, imagine that you are buried in the soil by your friends, and your head is deep below the ground surface. How do you describe the force intensity (i.e., stress) on your head (assuming your head is just like a particle in the sea of the soil)?

If the soil is replaced by still water, then the answer would be much simpler. The magnitude of the pressure (stress) on the top of your head would be the same as the pressure on your cheeks, and the direction of the pressure would always be perpendicular to the surface where the pressure applies. You've learned these in your high school. And you've learned that the magnitude of the pressure is $\sigma = \rho gh$, where ρ is the mass density of the water, g is the gravitational acceleration, and h is the depth of your head. In general, to describe the force intensity at a certain position in still water, we place an infinitesimally small body at that position, and measure the force per unit surface area on that body.

In the soil (which is a solid material rather than water), the behavior is quite different. First, the magnitude of the pressure on the top of your head may not be the same as that on your cheeks. Second, the direction of pressure is not necessarily perpendicular to the surface where the pressure applies. However, the above definition of stresses for water still holds (see [2]). ↩

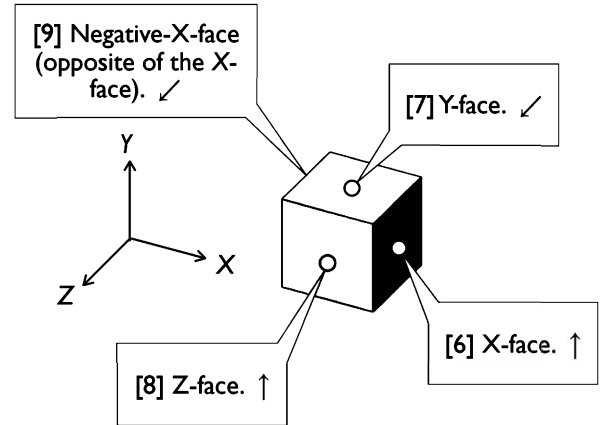
Definition of Stress

[2] The stress at a certain point can be defined as *the force per unit area acting on the boundary faces of an infinitesimally small body centered at that point* [3]. The small body can be any shape. In general, the stress values may be different at different faces, and the stress directions are not necessarily normal to the surfaces. And the small body can be any shape. To describe the stress in a systematic way, we usually use an infinitesimally small cube [4] of which each edge is parallel to a coordinate axis. If we can find the stresses on the faces of a small cube, we then can calculate the stresses on any faces of a small body of any shape. →



X-Face, Y-Face, and Z-Face

[5] Each of the six faces of the cube can be assigned an identifier, namely X-face, Y-face, Z-face, negative-X-face, negative-Y-face, and negative-Z-face, respectively [6-9]. Note that the outer normal of the X-face is in the positive direction of the X-axis, and so forth. →

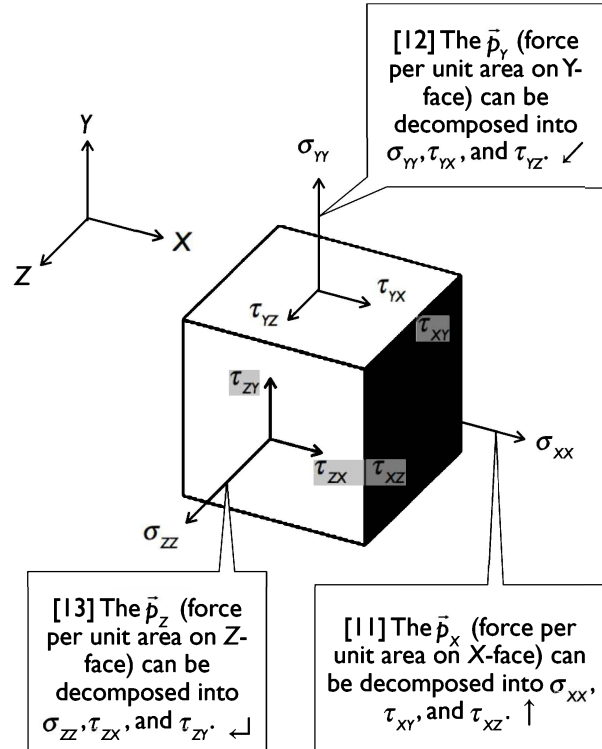


Stress Components

[10] Let \vec{p}_x be the force per unit area acting on the X-face. In general, \vec{p}_x may not be normal to the X-face. We may decompose \vec{p}_x into X-, Y-, and Z-component, and denote σ_{xx} , τ_{xy} , and τ_{xz} respectively [11]. The first subscript (X) is used to indicate the **face** on which the stress components act, while the second subscript (X, Y, or Z) is used to indicate the **direction** of the stress components. Note that σ_{xx} is normal to the face, while τ_{xy} and τ_{xz} are parallel to the face. Therefore, σ_{xx} is called a normal stress, while τ_{xy} and τ_{xz} are called shear stresses. We usually use the symbol σ for a normal stress and τ for a shear stress.

Similarly, let \vec{p}_y be the force per unit area acting on the Y-face and we may decompose \vec{p}_y into a normal component (σ_{yy}) and two shear components (τ_{yx} and τ_{yz}) [12]. Also, let \vec{p}_z be the force per unit area acting on the Z-face and we may decompose \vec{p}_z into a normal component (σ_{zz}) and two shear components (τ_{zx} and τ_{zy}) [13]. Organized in a matrix form, these stress components may be written as

$$\{\sigma\} = \begin{pmatrix} \sigma_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_{zz} \end{pmatrix} \quad (1) \rightarrow$$



Stress Components on Other Faces

[14] It can be proven that the stress components on the negative-X-face, negative-Y-face, and negative-Z-face can be derived from the 9 stress components in Eq. (1), last page. For example, on the negative-X-face, the stress components have exactly the same stress values as those on the X-face but with opposite directions [15]. Similarly, the stress components on the negative-Y-face have the same stress values as those on the Y-face but with opposite directions [16], and the stress components on the negative-Z-face have the same stress values as those on the Z-face but with opposite directions [17].

These can be proved by taking the cube as free body and applying the force equilibria in X, Y, and Z directions respectively.

On an arbitrary face (which may not be parallel or perpendicular to an axis), the stress components also can be calculated from the 9 stress components in Eq. (1), last page. This can be done by using a Mohr's circle (1.4.2[6], page 41). →

Symmetry of Shear Stresses

[18] It also can be proven that the shear stresses are symmetric; i.e.,

$$\tau_{xy} = \tau_{yx}, \quad \tau_{yz} = \tau_{zy}, \quad \tau_{zx} = \tau_{xz} \quad (2)$$

These can be proved by taking the cube as free body and applying the moment equilibria in X, Y, and Z directions respectively.

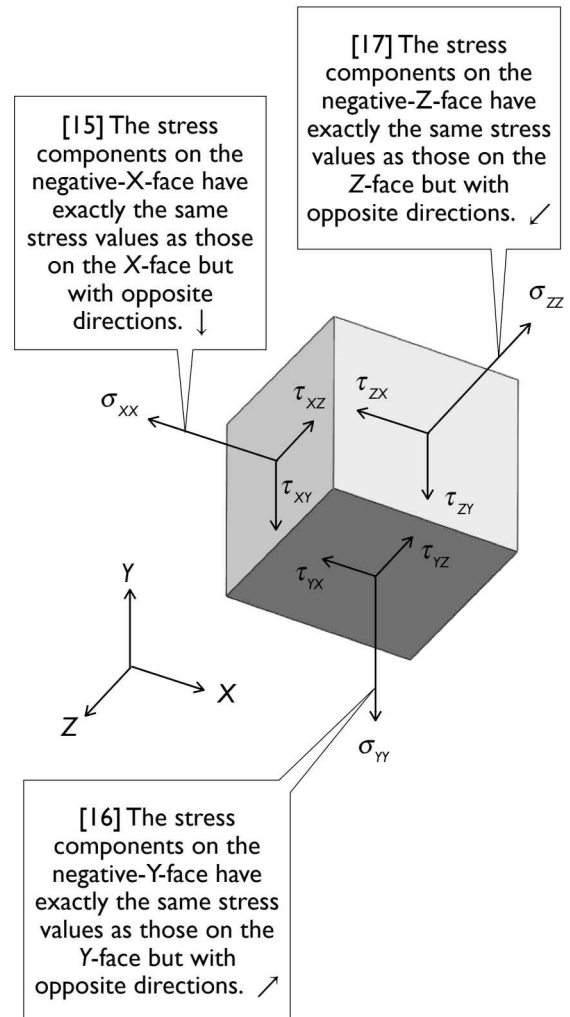
Stress State

We now conclude that 3 normal stress components and 3 shear stress components are needed to describe the **stress state** at a certain point; therefore, a stress state may be described using a vector

$$\{\sigma\} = \left\{ \sigma_x \quad \sigma_y \quad \sigma_z \quad \tau_{xy} \quad \tau_{yz} \quad \tau_{zx} \right\} \quad (3)$$

Note that, to be more concise, we use σ_x in place of σ_{xx} , σ_y in place of σ_{yy} , and σ_z in place of σ_{zz} .

These 6 components are, of course, functions of position. #



I.2.4 Strain

[1] Strains are quantities to describe how the material in a body is stretched and distorted. Strains are defined as the displacements of a point relative to its neighboring points. Although the notations of strain components are similar to those of stress components, the concepts of strains are even more difficult to comprehend.

Let's consider 2D cases first; the concepts can be extended to 3D cases. Consider a point A and its neighboring points B and C, which are respectively along X-axis and Y-axis [2]. Suppose that, after deformation, ABC moves to a new configuration A'B'C' [3]. Keep in mind that, in this discussion, we assume that the deformation is infinitesimally small. Under the small deformation assumption, the normal strains in X-axis and Y-axis can be defined respectively as

$$\varepsilon_x = \frac{A'B' - AB}{AB} \quad (\text{dimensionless}) \quad (1)$$

$$\varepsilon_y = \frac{A'C' - AC}{AC} \quad (\text{dimensionless}) \quad (2)$$

The strains defined in (1) and (2) represent the stretch at the point A in X-direction and Y-direction respectively. Stretch is not the only deformation modes; there are other deformation modes: changes of angles; e.g., from $\angle CAB$ to $\angle C'A'B'$, which is defined as the shear strain in XY-plane,

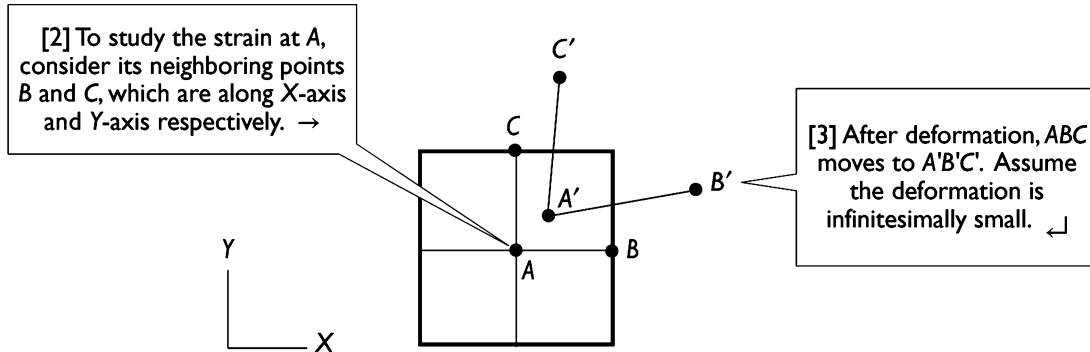
$$\gamma_{xy} = \angle CAB - \angle C'A'B' \quad (\text{rad}) \quad (3)$$

Note that the normal strains (1-2) and the shear strain (3) are all dimensionless, since the radian is also regarded as dimensionless.

In the above illustration, we consider only 2D cases. In general, the stretching may also occur in Z-direction and the shearing may also occur in YZ-plane and ZX-plane. Therefore, we need six strain components to completely describe the stretching and shearing of the material at a point:

$$\{\varepsilon\} = \left\{ \varepsilon_x \quad \varepsilon_y \quad \varepsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx} \right\} \quad (4)$$

These 6 components are, of course, functions of position. ✓



Why Are They Called Normal/Shear Strains?

[4] The definitions in Eqs. (1-3) do not explain why they are called "normal" strains and "shear" strain, respectively. To clarify this, let's redefine normal and shear strains using a different but equivalent way.

First, we translate and rotate $A'B'C'$ such that A' coincides with A and $A'C'$ aligns with AC [5]. Now the vector BB' is the "absolute" displacement (displacement excluding rigid body motion) of a neighboring point B which is on X -axis [6]. This displacement BB' can be decomposed into two components: BD and DB' ; the former is called the normal component, while the latter is called the shear component. They are so named because BD is normal to the X -face and DB' is parallel to the X -face. The normal strain and shear strain on X -face are then defined respectively by dividing the components with the original length,

$$\varepsilon_x = \frac{BD}{AB} \quad (\text{dimensionless}) \quad (5)$$

$$\gamma_{xy} = \frac{DB'}{AB} \quad (\text{rad}) \quad (6)$$

Note that, under the assumption of small deformation, the definition in Eq. (5) is the same as that in Eq. (1), while the definition in Eq. (6) is the same as that in Eq. (3). Also note that there are two subscripts in the shear strain γ_{xy} . The first subscript X is the face where the shearing occurs, while the second subscript Y is the direction of the shearing.

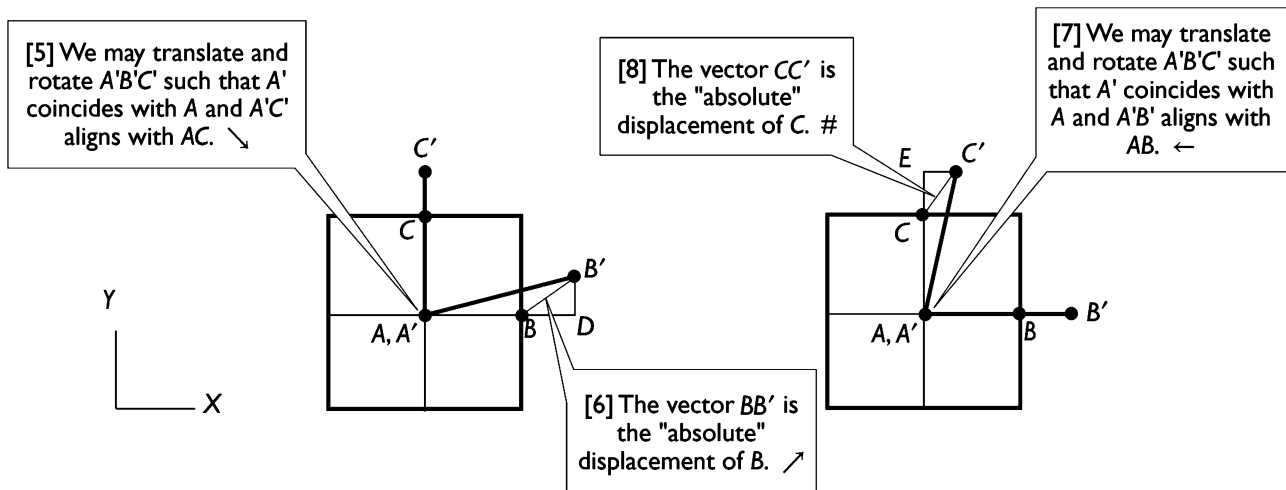
Similarly, we may translate and rotate $A'B'C'$ such that A' coincides with A and $A'B'$ aligns with AB [7]. Now the vector CC' is the "absolute" displacement of a neighboring point C which is on Y -axis [8]. This displacement CC' can be decomposed into two components: CE and EC' ; the former is the normal component, while the latter is the shear component. The normal strain and shear strain on Y -face is then defined by

$$\varepsilon_y = \frac{CE}{AC} \quad (\text{dimensionless}) \quad (7)$$

$$\gamma_{yx} = \frac{EC'}{AC} \quad (\text{rad}) \quad (8)$$

Note that, under the assumption of small deformation, the definition in Eq. (7) is the same as that in Eq. (2), while the definition in Eq. (8) is the same as that in Eq. (3). From Eqs. (3, 6, 8), we may write

$$\gamma_{xy} = \gamma_{yx} = \text{change of a right angle in } XY\text{-plane (rad)} \quad (9) \leftarrow$$



1.2.5 Governing Equations

[1] Let's summarize what we have concluded so far. In structural analysis, we can use the following quantities (all or part of them) to describe the response of a structure subject to environmental conditions:

$$\begin{aligned}\{u\} &= \begin{Bmatrix} u_x & u_y & u_z \end{Bmatrix} && \text{Copy of 1.2.2(1), page 24} \\ \{\sigma\} &= \begin{Bmatrix} \sigma_x & \sigma_y & \sigma_z & \tau_{xy} & \tau_{yz} & \tau_{zx} \end{Bmatrix} && \text{Copy of 1.2.3(3), page 27} \\ \{\varepsilon\} &= \begin{Bmatrix} \varepsilon_x & \varepsilon_y & \varepsilon_z & \gamma_{xy} & \gamma_{yz} & \gamma_{zx} \end{Bmatrix} && \text{Copy of 1.2.4(4), page 28}\end{aligned}$$

To solve for these 15 quantities, we must establish 15 equations. These equations are called governing equations; they govern the structure's behaviors. These equations, introduced in the rest of this section, include 3 equilibrium equations, 6 strain-displacement relations, and 6 stress-strain relations. #

1.2.6 Equilibrium Equations

[1] The stress components in Eq. 1.2.3(3) (page 27) must satisfy the principle of force equilibrium:

$$\sum F_x = 0, \quad \sum F_y = 0, \quad \sum F_z = 0 \quad (1)$$

If we apply Eqs. (1) on a point INSIDE the structural body, we can obtain three equilibrium equations involving the stress components:

$$\begin{aligned}\frac{\partial \sigma_x}{\partial X} + \frac{\partial \tau_{xy}}{\partial Y} + \frac{\partial \tau_{xz}}{\partial Z} + b_x &= 0 \\ \frac{\partial \tau_{yx}}{\partial X} + \frac{\partial \sigma_y}{\partial Y} + \frac{\partial \tau_{yz}}{\partial Z} + b_y &= 0 \\ \frac{\partial \tau_{zx}}{\partial X} + \frac{\partial \tau_{zy}}{\partial Y} + \frac{\partial \sigma_z}{\partial Z} + b_z &= 0\end{aligned} \quad (2)$$

where b_x, b_y, b_z are components of body forces (forces distributed in the body, with SI unit N/m^3). If we apply Eqs. (1) on a point ON the boundary surface of the structural body, the three equilibrium equations will have the form:

$$\begin{aligned}\sigma_x n_x + \tau_{xy} n_y + \tau_{xz} n_z + S_x &= 0 \\ \tau_{yx} n_x + \sigma_y n_y + \tau_{yz} n_z + S_y &= 0 \\ \tau_{zx} n_x + \tau_{zy} n_y + \sigma_z n_z + S_z &= 0\end{aligned} \quad (3)$$

where S_x, S_y, S_z are components of surface forces (forces distributed on the boundary, with SI unit N/m^2), and n_x, n_y, n_z are components of the unit normal vector on the boundary surface.

We will not discuss the derivations of Eqs. (2-3); they can be found in any Solid Mechanics textbooks^[Refs 1,2]. Here, we just reiterate that the equilibrium equations originate from Eq. (1). Also note that, in order to derive Eq. (2), the stress components in 1.2.3[11-13, 15-17] (pages 26-27) must be expanded to include first differential terms. #

1.2.7 Strain-Displacement Relations

[1] There exist geometric relations between the displacement and the strain. Under the assumption of small deformation, the relations are linear:

$$\begin{aligned}\varepsilon_x &= \frac{\partial u_x}{\partial X}, \quad \varepsilon_y = \frac{\partial u_y}{\partial Y}, \quad \varepsilon_z = \frac{\partial u_z}{\partial Z} \\ \gamma_{xy} &= \frac{\partial u_x}{\partial Y} + \frac{\partial u_y}{\partial X}, \quad \gamma_{yz} = \frac{\partial u_y}{\partial Z} + \frac{\partial u_z}{\partial Y}, \quad \gamma_{zx} = \frac{\partial u_z}{\partial X} + \frac{\partial u_x}{\partial Z}\end{aligned}\quad (1)$$

Again, we will not discuss the derivations of Eq. (1); they can be found in any Solid Mechanics textbooks^[Refs 1, 2]. Here, we just want to reiterate that Eq. (1) is derived from mathematics (geometry) without applying any physical principles. Second, if the assumption of small deformation is removed, a strain-displacement relation still exists but is no longer linear; there will be some high-order differential terms in the equations. #

1.2.8 Stress-Strain Relations

[1] With 3 equilibrium equations (Eqs. 1.2.6(2 or 3), last page) and 6 strain-displacement relations--Eq. 1.2.7(1)--we need 6 additional equations to solve the governing equations. From an engineering point of view, it is practical to assume a relation between stress and strain. Experiment data show that a linear relation between stress and strain often can be adopted; it is called Hooke's law^[Refs 1, 2, 3]:

$$\begin{aligned}\varepsilon_x &= \frac{\sigma_x}{E} - \nu \frac{\sigma_y}{E} - \nu \frac{\sigma_z}{E} \\ \varepsilon_y &= \frac{\sigma_y}{E} - \nu \frac{\sigma_z}{E} - \nu \frac{\sigma_x}{E} \\ \varepsilon_z &= \frac{\sigma_z}{E} - \nu \frac{\sigma_x}{E} - \nu \frac{\sigma_y}{E} \\ \gamma_{xy} &= \frac{\tau_{xy}}{G}, \quad \gamma_{yz} = \frac{\tau_{yz}}{G}, \quad \gamma_{zx} = \frac{\tau_{zx}}{G}\end{aligned}\quad (1)$$

Although Eq. (1) is purely an assumption (no physical or mathematical principles applied), it is proved to be very useful and satisfactory in many cases, depending on the material and the application. Eq. (1) is called a *material model*; it characterizes the behavior of a material, independent of the geometry and environmental conditions.

There are three material parameters in Eq. (1): the Young's modulus E , the Poisson's ratio ν , and the shear modulus G . In SI, the Young's modulus and the shear modulus have a unit of pascal (Pa) and the Poisson's ratio is dimensionless. It can be shown that these three quantities are not independent to each other; they satisfy the relation^[Ref 3]

$$G = \frac{E}{2(1 + \nu)} \quad (2)$$

We conclude that, for an isotropic, linearly elastic material, any two of E , ν , and G can be used to describe the stress-strain relation. In Workbench, we usually input the Young's modulus and the Poisson's ratio to define an **Isotropic Elasticity** model (1.1.3[5-6], page 12). ↵

Thermal Effects

[2] Consider that the temperature changes over the structural body. Since a temperature change ΔT induces a strain $\alpha\Delta T$, in which α is the *coefficient of thermal expansion*, this *thermal strain* should be added to Eq. (1) (last page); i.e.,

$$\begin{aligned}
 \varepsilon_x &= \frac{\sigma_x}{E} - \nu \frac{\sigma_y}{E} - \nu \frac{\sigma_z}{E} + \alpha\Delta T \\
 \varepsilon_y &= \frac{\sigma_y}{E} - \nu \frac{\sigma_z}{E} - \nu \frac{\sigma_x}{E} + \alpha\Delta T \\
 \varepsilon_z &= \frac{\sigma_z}{E} - \nu \frac{\sigma_x}{E} - \nu \frac{\sigma_y}{E} + \alpha\Delta T \\
 \gamma_{xy} &= \frac{\tau_{xy}}{G}, \quad \gamma_{yz} = \frac{\tau_{yz}}{G}, \quad \gamma_{zx} = \frac{\tau_{zx}}{G}
 \end{aligned} \tag{3}$$

Orthotropic Elasticity

For orthotropic materials ([4.1.1 [3], page 527), in which there exist three mutual orthogonal planes of material symmetry, Hooke's law can be generalized to^[Refs 1, 2, 3]

$$\begin{aligned}
 \varepsilon_x &= \frac{\sigma_x}{E_x} - \nu_{yx} \frac{\sigma_y}{E_y} - \nu_{zx} \frac{\sigma_z}{E_z} + \alpha_x \Delta T \\
 \varepsilon_y &= \frac{\sigma_y}{E_y} - \nu_{zy} \frac{\sigma_z}{E_z} - \nu_{xy} \frac{\sigma_x}{E_x} + \alpha_y \Delta T \\
 \varepsilon_z &= \frac{\sigma_z}{E_z} - \nu_{xz} \frac{\sigma_x}{E_x} - \nu_{yz} \frac{\sigma_y}{E_y} + \alpha_z \Delta T \\
 \gamma_{xy} &= \frac{\tau_{xy}}{G_{xy}}, \quad \gamma_{yz} = \frac{\tau_{yz}}{G_{yz}}, \quad \gamma_{zx} = \frac{\tau_{zx}}{G_{zx}}
 \end{aligned} \tag{4}$$

where E_x, E_y, E_z are Young's moduli in their respective directions, G_{xy}, G_{yz}, G_{zx} are the shear moduli in their respective planes, and $\nu_{xy}, \nu_{yz}, \nu_{zx}$ are the Poisson's ratios in their respective planes. The first subscript in each of the Poisson's ratios refers to the direction of the load, and the second to the direction of the contraction. For example, ν_{xy} represents the amount of contraction in Y-direction, when the material is stretched in X-direction. #

1.2.9 Summary

[1] We now have 15 equations, including three equilibrium equations, either (INSIDE the structural body)

$$\frac{\partial \sigma_x}{\partial X} + \frac{\partial \tau_{xy}}{\partial Y} + \frac{\partial \tau_{xz}}{\partial Z} + b_x = 0$$

$$\frac{\partial \tau_{yx}}{\partial X} + \frac{\partial \sigma_y}{\partial Y} + \frac{\partial \tau_{yz}}{\partial Z} + b_y = 0$$

$$\frac{\partial \tau_{zx}}{\partial X} + \frac{\partial \tau_{zy}}{\partial Y} + \frac{\partial \sigma_z}{\partial Z} + b_z = 0$$

Copy of 1.2.6(2), page 30

or (ON the boundary surface)

$$\sigma_x n_x + \tau_{xy} n_y + \tau_{xz} n_z + S_x = 0$$

$$\tau_{yx} n_x + \sigma_y n_y + \tau_{yz} n_z + S_y = 0$$

$$\tau_{zx} n_x + \tau_{zy} n_y + \sigma_z n_z + S_z = 0$$

Copy of 1.2.6(3), page 30

and six equations describing the strain-displacement relation

$$\varepsilon_x = \frac{\partial u_x}{\partial X}, \quad \varepsilon_y = \frac{\partial u_y}{\partial Y}, \quad \varepsilon_z = \frac{\partial u_z}{\partial Z}$$

$$\gamma_{xy} = \frac{\partial u_x}{\partial Y} + \frac{\partial u_y}{\partial X}, \quad \gamma_{yz} = \frac{\partial u_y}{\partial Z} + \frac{\partial u_z}{\partial Y}, \quad \gamma_{zx} = \frac{\partial u_z}{\partial X} + \frac{\partial u_x}{\partial Z}$$

Copy of 1.2.7(1), page 31

and six equations describing the stress-strain relation

$$\varepsilon_x = \frac{\sigma_x}{E} - \nu \frac{\sigma_y}{E} - \nu \frac{\sigma_z}{E} + \alpha \Delta T$$

$$\varepsilon_y = \frac{\sigma_y}{E} - \nu \frac{\sigma_z}{E} - \nu \frac{\sigma_x}{E} + \alpha \Delta T$$

$$\varepsilon_z = \frac{\sigma_z}{E} - \nu \frac{\sigma_x}{E} - \nu \frac{\sigma_y}{E} + \alpha \Delta T$$

$$\gamma_{xy} = \frac{\tau_{xy}}{G}, \quad \gamma_{yz} = \frac{\tau_{yz}}{G}, \quad \gamma_{zx} = \frac{\tau_{zx}}{G}$$

Copy of 1.2.8(3), page 32

In theory, these 15 equations can be solved for 15 quantities:

$$\{u\} = \left\{ u_x \quad u_y \quad u_z \right\}$$

Copy of 1.2.2(1), page 24

$$\{\sigma\} = \left\{ \sigma_x \quad \sigma_y \quad \sigma_z \quad \tau_{xy} \quad \tau_{yz} \quad \tau_{zx} \right\}$$

Copy of 1.2.3(3), page 27

$$\{\varepsilon\} = \left\{ \varepsilon_x \quad \varepsilon_y \quad \varepsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx} \right\}$$

Copy of 1.2.4(4), page 28

In practice, only a few simple "textbook problems" can be solved analytically. Most real-world problems are too complicated to solve analytically. The complexity mostly comes from the geometry and the environmental conditions. Numerical methods are usually the only feasible methods. The finite element methods, which have been the most successful numerical methods for *boundary-value problems* (such as the problems described in this section), are implemented in ANSYS Workbench to solve the governing equations. ↵

Remark

[2] By "governing equations" of a structural analysis problem, we sometimes mean the equilibrium equations: Eq. 1.2.6(2) (page 30) governs the behavior in the body and Eq. 1.2.6(3) (page 30) governs the behavior on the boundary. The two sets of equations constitute a boundary value problem.

It is possible to replace the stress components in the equilibrium equations by strain components using Eq. 1.2.8(3) (page 32), and in turn replace the strain components by displacement components using Eq. 1.2.7(1) (page 31). The result is a set of three equilibrium equations involving three displacement components, and we can solve the three differential equations for the three displacement components.

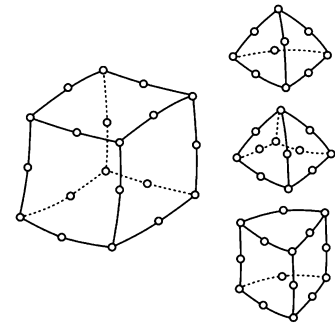
That is how ANSYS Workbench solves a structural problem (see Eq. 1.3.1(1), next page). #

References

1. Cook, R. D. and Young, W. C., *Advanced Mechanics of Materials*, Macmillan, 1985.
2. Haslach, H. W. Jr. and Armstrong, R. W., *Deformable Bodies and Their Material Behavior*, John Wiley & Sons, Inc., 2004; Chapter 9 Plasticity.
3. Beer, F. P., Johnston, E. R. Jr., and DeWolf, J. T., *Mechanics of Materials*, 3rd Ed., McGraw Hill, 2002.

Section 1.3

Finite Element Methods: A Concise Introduction



This section introduces (a) terminology relevant to finite element methods, such as degrees of freedom, shape functions, stiffness matrix, etc., that will be used throughout the book and (b) the basic procedure of finite element methods, so the students have a better understanding about how the Workbench performs a simulation.

1.3.1 Basic Procedure

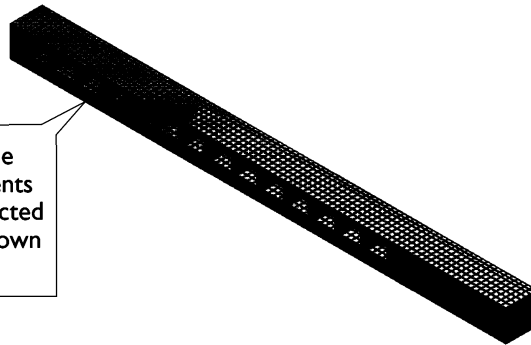
[1] As mentioned (at the bottom of 1.2.9[1], page 33), most real-world problems are too complicated to be solved analytically, because of the complexity of geometry and/or environmental conditions. Further, when nonlinearity and dynamic effects are considered in the problems, then their analytical solutions are practically unreachable.

A basic idea of finite element methods is to divide the structural body into many small and geometrically simple bodies, called *elements*, so the equations of each element can be established, and all the equations are then solved simultaneously. The elements have finite sizes (in contrast to the infinitesimal sizes of elements in Calculus), thus the name *Finite Element Methods*.

The elements are assumed to be connected by *nodes* located at the elements' edges and vertices. An additional idea is to solve unknown discrete values (e.g., displacements at the nodes) rather than to solve unknown functions (e.g., displacement fields over the body). Since the displacement on each node is a vector and has three components (in 3D cases), the number of total unknown quantities to be solved is three times the number of nodes [2].

The types of elements available in the ANSYS Workbench and their specific configuration of nodes will be given in 1.3.3, pages 37-39. ↓

[2] In the case of the pneumatic finger, the structural body is divided into 3,395 elements (1.1.5[6], page 16). The elements are connected by 19,201 nodes. There are $3 \times 19,201$ unknown displacement values to be solved. ↓



[3] The nodal displacement components, collectively denoted by a vector $\{D\}$, are called the *degrees of freedom* (DOFs) of the structure. They are so called because these values fully define the response of a structure. In a static case, the system of equilibrium equations has the following form (also see 1.2.9[2], last page)

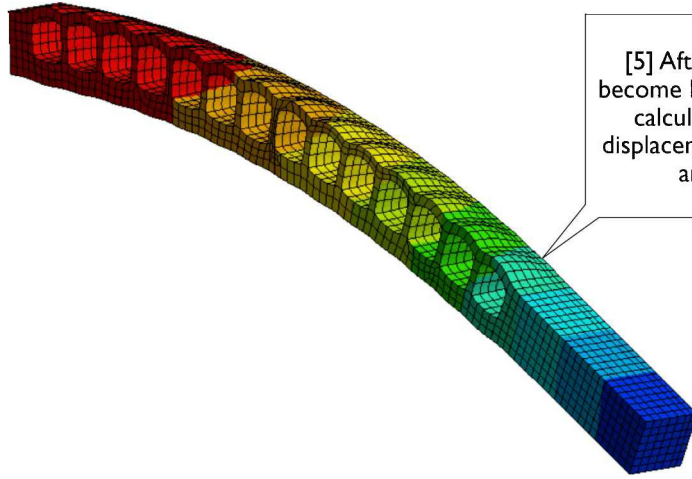
$$[K]\{D\} = \{F\} \quad (1) \quad \leftarrow$$

[4] The size of Eq. (1) (last page) is determined by the number of degrees of freedom, which is, in 3D cases, three times the number of nodes. The vector $\{F\}$ is the external forces, which is calculated from the environmental conditions. Physical meaning of the matrix $[K]$ can be understood by thinking of the structure as a spring, $\{F\}$ as external force, and $\{D\}$ as the deformation of the spring. Then $[K]$ is the spring constant, or the stiffness of the spring. In general multiple-degrees-of-freedom cases, $[K]$ is called the *stiffness matrix* of the structure, and the physical meaning of the i^{th} column is the forces required to make the i^{th} DOF a unit displacement while restraining the other DOFs from any displacements.

Note that, for a linear structure, $[K]$ is a constant matrix; while for nonlinear cases (1.1.12, pages 22-23), $[K]$ may be viewed as a function of $\{D\}$, which will be discussed in Chapters 13 and 14. For dynamic cases (1.1.10 and 1.1.11, pages 21-22), dynamic effects need to be added to Eq. (1) (last page), which will be discussed in Chapters 11 and 12.

After the discrete nodal displacements $\{D\}$ in Eq. (1) are solved, the displacement fields $\{u\}$ are calculated by interpolating the nodal displacements, either linearly or quadratically [5] (also see Eq. 1.3.2(2), this page). These interpolating functions are called *shape functions*. The concepts of shape functions are crucial in the finite element methods; they will be discussed further in 1.3.2.

As soon as the displacement fields $\{u\}$ become known, the strain fields can be calculated by Eq. 1.2.7(1) (page 31), the strain-displacement relation. The stress fields in turn can be calculated by Eq. 1.2.8(3) (page 32), the stress-strain relation. →



[5] After discrete nodal displacements become known, the displacement fields are calculated by interpolating the nodal displacements. The interpolating functions are called *shape functions*. #

1.3.2 Shape Functions and the Order of Element

[1] As mentioned (1.3.1[4-5], this page), the displacement fields $\{u\}$ are calculated by interpolating the nodal displacements. The interpolating functions are called shape functions; the shape functions establish a relation between the displacement fields and the nodal displacements.

As an example, consider a 2D 4-node quadrilateral element ([3], next page). The nodal displacements of the element, collectively denoted by a vector $\{d\}$, have 8 components

$$\{d\} = \{ d_1 \ d_2 \ d_3 \ d_4 \ d_5 \ d_6 \ d_7 \ d_8 \} \quad (1)$$

The displacement fields $\{u\}$ can be calculated by interpolating the nodal displacements $\{d\}$

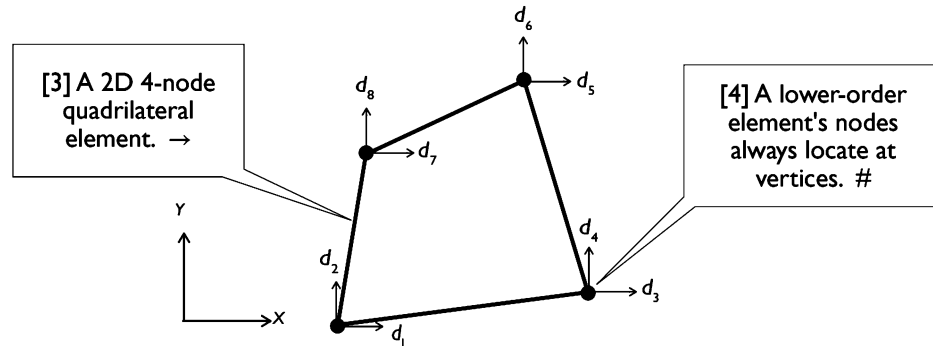
$$\{u\} = [N]\{d\} \quad (2) \quad \leftarrow$$

[2] In Eq. (2), last page, $[N]$ is the *matrix of shape functions*. The role of the shape functions is the interpolating functions from the nodal displacements $\{d\}$ to the displacement fields $\{u\}$. Note that the components of nodal displacements $\{d\}$ are discrete values and the components of displacement fields $\{u\}$ are continuous functions of (X, Y, Z) . The shape functions in $[N]$ are the bridge between the continuous functions $\{u\}$ and the discrete values $\{d\}$.

Now, let's examine the dimension of the $[N]$ matrix using this example. Since $\{d\}$ has 8 components and $\{u\}$ has 3 components (Eq. 1.2.2(1), page 24), the matrix $[N]$ must be of dimension 3×8 .

Since $\{u\}$ contains functions of (X, Y, Z) and $\{d\}$ contains discrete values, $[N]$ must contain functions of (X, Y, Z) . Besides, for the element shown in [3], the interpolating points are at the vertices of the element, so the shape functions must be a linear form. When the shape functions are linear, the element is called a *linear element*, *first-order element*, or *lower-order element* [4].

Often, using quadratic polynomials as shape functions can be more efficient. In such cases, a node is added on the middle of each edge of the element; the added nodes are called the *midside nodes*; the element is called a *quadratic element*, *second-order element*, or *higher-order element*. In this book, we will not use the term *linear element*, to avoid confusing it with the terms such as linear material. ↓



1.3.3 Workbench Element Types

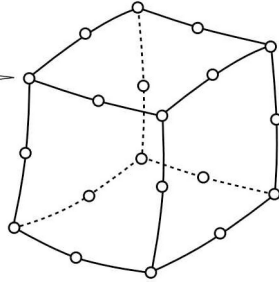
[1] ANSYS literally provides hundreds of element types^[Ref 1]. By default, Workbench automatically chooses appropriate element types during the meshing process^[Ref 2]; to select a specific type of element, you need to use APDL. To identify these element types, each element type is assigned a code name (e.g., SOLID186, PLANE183, etc.).

For 2D solid-body cases, the element shapes available are quadrilateral (4-sided) and triangular (3-sided). For 3D solid-body cases, the available shapes are hexahedral (6-faced), triangle-based prism (5-faced), quadrilateral-based pyramid (5-faced), and tetrahedral (4-faced). As mentioned, Workbench chooses element types according to the types of the structural bodies. Currently Workbench supports 4 body types: 2D solid body, 3D solid body, 3D surface body, and 3D line body. 3D surface bodies are geometrically 2D but spatially 3D, while 3D line bodies are geometrically 1D but spatially 3D. How Workbench chooses an element type according to the body type is illustrated as follows.

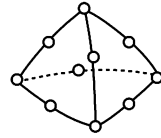
3D Solid Bodies

Workbench meshes a 3D solid body with SOLID186^[Ref 3], a 3D 20-node second-order structural solid element ([2], next page). The element is originally hexahedral. By combining some of the nodes, the element can degenerate to a triangle-based prism [3], quadrilateral-based pyramid [4], or tetrahedron [5]. Degeneration is useful since it allows different shapes of elements to be mixed up in a body. If a body is to be meshed with tetrahedral elements exclusively, Workbench meshes the body with SOLID187^[Ref 4], which is a 3D 10-node tetrahedral second-order structural solid element and has a shape the same as that in [5]. Workbench allows an option to drop off elements' midside nodes; in that case the edges become straight and the element becomes first-order. ↵

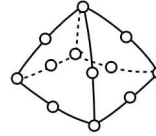
[2] Hexahedron, or 3D 20-node structural solid. Each node has 3 translational degrees of freedom: D_x , D_y , and D_z . →



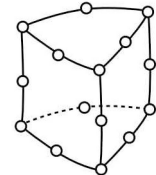
[5] Tetrahedron. ✓



[4] Quadrilateral-based pyramid. ↑



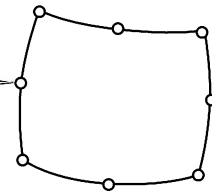
[3] Triangle-based prism. ↑



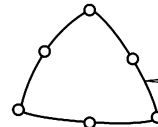
2D Solid Bodies

[6] Workbench meshes a 2D solid body with PLANE183^[Ref 5], a 2D 8-node second-order structural solid element [7]. The element is originally quadrilateral. By combining some of the nodes, the element can degenerate to a triangle [8]. If you choose to drop midside nodes, the edges become straight, and the Workbench meshes the body with PLANE182^[Ref 6], a 2D 4-node first-order structural solid element. It is important to remember that all 2D solid elements must be arranged on **XYPlane**. ↓

[7] Quadrilateral, or 2D 8-node structural solid. Each node has 2 translational degrees of freedom: D_x and D_y . →



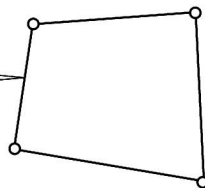
[8] Triangle. ↓



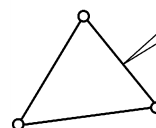
3D Surface Bodies

[9] Workbench meshes a 3D surface body with SHELL181^[Ref 7], a 3D 4-node first-order structural shell element [10], or SHELL281^[Ref 8], a 3D 8-node second-order structural shell. The element is originally quadrilateral but can degenerate to a triangle [11]. ↓

[10] Quadrilateral shell, or 3D 4-node structural shell. Each node has 3 translational and 3 rotational degrees of freedom: D_x , D_y , D_z , R_x , R_y , and R_z . →

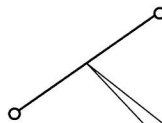


[11] Triangular shell. ←



3D Line Bodies

[12] For a 3D line body, the Workbench meshes it with BEAM188^[Ref 9], a 3D 2-node first-order beam element [13]. ↓



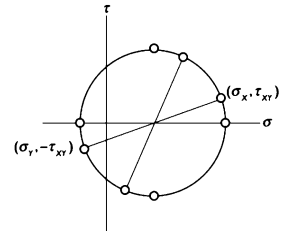
[13] 3D 2-Node beam. Each node has 3 translational and 3 rotational degrees of freedom: $D_x, D_y, D_z, R_x, R_y, \text{ and } R_z$. #

References

1. All Help>Mechanical APDL>Theory Reference>13. Element Library
2. The following page is available in version 13 and earlier but removed since version 14.
ANSYS Help System//Mechanical APDL//Theory Reference//1.4.1. Elements Used by the ANSYS Workbench Product
3. All Help>Mechanical APDL>Theory Reference>13.186. SOLID186
4. All Help>Mechanical APDL>Theory Reference>13.187. SOLID187
5. All Help>Mechanical APDL>Theory Reference>13.183. PLANE183
6. All Help>Mechanical APDL>Theory Reference>13.182. PLANE182
7. All Help>Mechanical APDL>Theory Reference>13.181. SHELL181
8. All Help>Mechanical APDL>Theory Reference>13.281. SHELL281
9. All Help>Mechanical APDL>Theory Reference>13.188. BEAM188

Section 1.4

Failure Criteria of Materials



Achieving functionality, safety, and reliability are the main purposes of structural simulations. Deformation usually relates to the functionality, while stress to the safety and reliability. Which stress should we look into to ensure that a structure doesn't fail? Normal stress? Shear stress? Or something else? How can we say a stress value is too large? What critical values should the stresses be compared with? In short, what are the failure criteria of materials? This section intends to answer these questions.

1.4.1 Ductile versus Brittle Materials

[1] Stress-strain relation is the most important characteristic of a material. We usually obtain a stress-strain relation by conducting a uniaxial tensile test. Two examples of stress-strain relation from uniaxial tensile tests are shown in [2-3]. In [2], the material exhibits a large amount of strain before it fractures [4]; it is called a *ductile material*. In [3], the material's fracture strain is relatively small [5]; it is called a *brittle material*. Fracture strain is a measure of ductility. There are essential differences between these two types of materials.

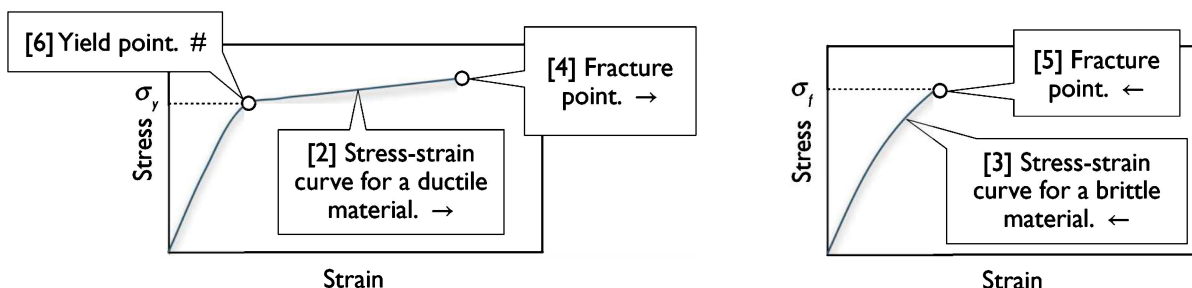
Failure Points: Yield Point or Fracture Point?

Mild steel is a typical ductile material. For ductile materials, there often exists an obvious yield point [6], beyond which the deformation would be too large, and the material is no longer reliable or functional; the failure is accompanied by excess deformation. Therefore, for ductile materials, we are concerned about whether the material reaches the yield point [6]; its corresponding stress is called the yield stress σ_y , which is the critical stress we want to compare with. But, with which stress to compare? σ_x ? σ_y ? σ_z ? τ_{xy} ? τ_{yz} ? τ_{zx} ? Or something else? 1.4.4 and 1.4.5 (pages 42-46) will answer this question.

Cast iron and ceramics are two examples of brittle materials. For a brittle material, there usually doesn't exist an obvious yield point, and we are concerned about its fracture point [5]; its corresponding stress is called the fracture stress σ_f , which is the critical stress we want to compare with. But, again, with which stress to compare? σ_x ? σ_y ? σ_z ? τ_{xy} ? τ_{yz} ? τ_{zx} ? Or something else? 1.4.3 (page 42) will answer this question.

Failure Modes: Tensile Failure or Shear Failure?

The fracture of brittle materials is mostly due to *tensile failure*; the yielding of ductile materials is mostly due to *shear failure* [Refs 1, 2]. The tensile failure of brittle materials is easy to observe: the failure always occurs after cracking, induced by tensile stresses. The shear failure of ductile materials can be observed in a standard uniaxial tensile test, in which the failure is accompanied by a necking phenomenon and a cone-shape breaking surface. It is important to remember that a material often fails due to a mix-up of these two mechanisms. ↓



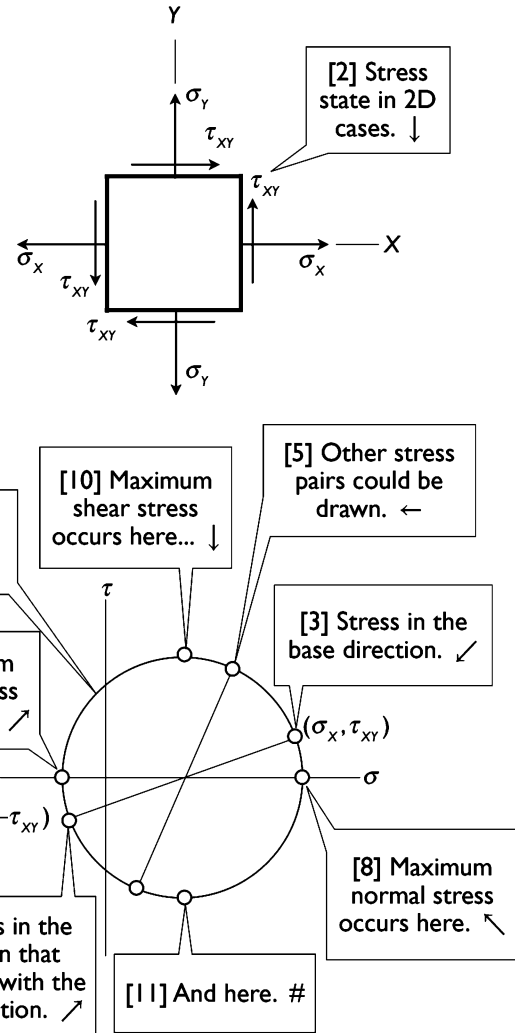
1.4.2 Principal Stresses

[1] We mentioned that, at a certain point, the stresses are different in different faces (or directions, 1.2.3[2], page 26). It then naturally raises a question: In which face does the normal stress reach its maximum? And in which face does the shear stress reach its maximum?

To present the concepts efficiently, consider a 2D case [2], and let's denote the X-direction the base direction. In the X-face, the stress can be expressed with a stress pair (σ_x, τ_{xy}) . Likewise, in an arbitrary face, the stress can be expressed with a stress pair (σ, τ) . Let's now try to find a relationship between normal stress σ and shear stress τ : how does σ vary with τ ?

First, we mark the stress pair (σ_x, τ_{xy}) in the σ - τ space [3]. Second, noting that the stress $(\sigma_y, -\tau_{xy})$ is also a stress pair, whose direction forms 90° (counter-clockwise) with the base direction, we mark the stress pair $(\sigma_y, -\tau_{xy})$ in the σ - τ space [4]. Similarly, we could draw other stress pairs in the σ - τ space [5]. The collection of these points forms a circle in the σ - τ space [6], called a *Mohr's circle*. The details can be found in any textbook of Solid Mechanics^[Refs 2, 3].

Here, we want to emphasize a concept: a *Mohr's circle* represents a stress state. With this useful concept, finding the maximum normal stress and maximum shear stress becomes straightforward. ↗



[7] A stress state defines a Mohr's circle, and vice versa. Further, the stress values, σ_x , σ_y , and τ_{xy} , fully define a Mohr's circle. Once we have a Mohr's circle, the points of maximum normal stress, minimum normal stress, and maximum shear stress can be located. The maximum normal stress is located at the right quarter-point of the Mohr's circle [8]. The minimum normal stress is located at the left quarter-point of the Mohr's circle [9]. The *maximum shear stress* is located at either the upper quarter-point or the lower quarter-point of the Mohr's circle [10-11]. Note that at the points of maximum and minimum normal stresses [8-9], the shear stresses vanish.

The maximum normal stress [8] is called the *maximum principal stress* and is denoted by σ_1 ; the minimum normal stress [9] is called the *minimum principal stress* and is denoted by σ_3 . Their corresponding directions are called *principal direction*. At a point of a 3D solid, there are three principal directions (and thus three principal stresses). The *medium principal stress* is denoted by σ_2 . In our example, the maximum principal stress [8] is a positive value, a tension; the minimum principal stress [9] is a negative value, a compression.

Given three stress values, σ_x , σ_y , and τ_{xy} , to define a Mohr's circle, we can easily calculate the values and their corresponding directions of the principal stresses and the maximum shear stress. We will not derive these formulas in this book; Workbench can report for you at your request. Finally, make sure you understand these concepts and can generalize the concepts to 3D cases. ↗

1.4.3 Failure Criterion for Brittle Materials

[1] As mentioned, the failure of brittle materials is a tensile failure (1.4.1[1], page 40). In other words, a brittle material fractures because its tensile stress reaches the fracture strength σ_f (1.4.1[5], page 40). Thus, we may state a failure criterion for brittle materials as follows: At a certain point of a body, if the maximum principal stress reaches the fracture strength of the material, it will fail. In short, a point of material fails if

$$\sigma_1 \geq \sigma_f \quad (1) \quad \#$$

1.4.4 Tresca Criterion for Ductile Materials

[1] As mentioned, the failure of ductile materials is a shear failure (1.4.1[1], page 40). In other words, a ductile material yields because its shear stress reaches the shear strength τ_y . Thus, we may state a failure criterion for ductile materials as follows: At a certain point of a body, if the maximum shear stress reaches the shear strength of the material, it will fail. In short, a point of material fails if

$$\tau_{\max} \geq \tau_y \quad (1)$$

The maximum shear stress τ_{\max} --from the geometry of a Mohr's circle (1.4.2[6], page 41)--is simply the radius of the circle. Noting that the diameter of the circle is $(\sigma_1 - \sigma_3)$, we may write down

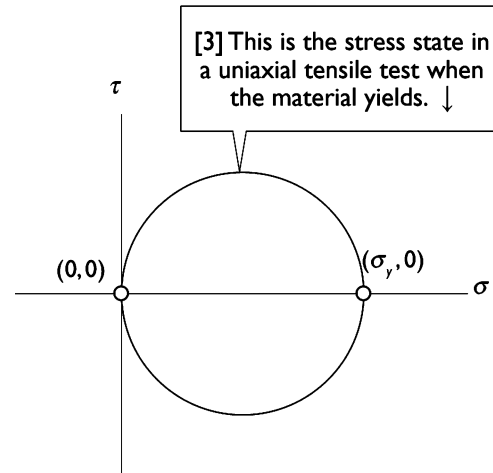
$$\tau_{\max} = \frac{\sigma_1 - \sigma_3}{2} \quad (2) \quad \checkmark$$

[2] In a uniaxial tensile test, the material yields when undergoing its yield stress σ_y in the axial direction. Let the axial direction be X-direction, then the stress state $(\sigma_x, \sigma_y, \tau_{xy})$ is $(\sigma_y, 0, 0)$ [3]; its maximum principal stress is σ_y while the minimum principal stress is zero. Thus, when the material yields, its shear stress is

$$\tau_y = \frac{\sigma_y}{2} \quad (3)$$

Substituting Eqs. (2-3) into (1), we have the criterion

$$\frac{\sigma_1 - \sigma_3}{2} \geq \frac{\sigma_y}{2} \quad (4) \quad \rightarrow$$



[4] Simplifying Eq. (4), we reach a conclusion that the material yields if

$$\sigma_1 - \sigma_3 \geq \sigma_y \quad (5)$$

In ANSYS, the quantity on the left-hand side, $(\sigma_1 - \sigma_3)$, is called the *stress intensity*, which the Workbench can report for you at your request. This failure criterion is called the *maximum shear stress criterion*, or *Tresca Criterion*, first proposed by Henri Tresca (1814-1885), a French mechanical engineer, in 1864. #

1.4.5 Von Mises Criterion for Ductile Materials

[1] The Tresca criterion (Eq. 1.4.4(5), last page) is simple but often not good enough for predicting the yielding of many ductile materials, particularly metals. The theory discussed in 1.4.4 (last page) is too simplified and may deviate from real-world situations. For example, the medium principal stress σ_2 plays no role in Eq. 1.4.4(5), but that is not always true. A more sophisticated theory, called the *von Mises criterion*, often predicts yielding more satisfactorily than Tresca criterion.

Hydrostatic Stress and Deviatoric Stress

Without loss of generality, a stress state

$$\{\sigma\} = \begin{Bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{Bmatrix}$$

can be expressed in a form of principal stresses if we choose the principal axes as coordinate axes,

$$\{\sigma\} = \begin{Bmatrix} \sigma_1 & 0 & 0 \\ 0 & \sigma_2 & 0 \\ 0 & 0 & \sigma_3 \end{Bmatrix} \quad (1)$$

Note that, as mentioned, in the principal directions, all the shear stresses vanish (1.4.2[7], page 41).

Define the hydrostatic stress as the average of the normal stresses,

$$p = \frac{\sigma_1 + \sigma_2 + \sigma_3}{3} \quad (2)$$

The stress state (1) can be decomposed into two parts

$$\{\sigma\} = \begin{Bmatrix} \sigma_1 & 0 & 0 \\ 0 & \sigma_2 & 0 \\ 0 & 0 & \sigma_3 \end{Bmatrix} = \begin{Bmatrix} p & 0 & 0 \\ 0 & p & 0 \\ 0 & 0 & p \end{Bmatrix} + \begin{Bmatrix} \sigma_1 - p & 0 & 0 \\ 0 & \sigma_2 - p & 0 \\ 0 & 0 & \sigma_3 - p \end{Bmatrix} \quad (3)$$

or, written in a more compact form,

$$\{\sigma\} = \{\sigma^p\} + \{\sigma^d\} \quad (4)$$

The first part $\{\sigma^p\}$ is called the *hydrostatic stress*, and the second part $\{\sigma^d\}$ is called the *deviatoric stress*, the stress deviating from the hydrostatic stress.

The deformation of a body can be thought of as a superposition of a dilation (volumetric change) and a distortion. The hydrostatic stress contributes exclusively to the dilation, while the deviatoric stress contributes exclusively to the distortion. The dilation plays no role in shear failure; it is the distortion that causes shear failure. In other words, the deviatoric stress

$$\{\sigma^d\} = \begin{Bmatrix} \sigma_1 - p & 0 & 0 \\ 0 & \sigma_2 - p & 0 \\ 0 & 0 & \sigma_3 - p \end{Bmatrix} \quad (5)$$

can be used to establish a shear failure criterion. However, it is not a scalar value. How can we use it to compare with a uniaxial yield stress σ_y ? We need a more elaborate theory to derive a useful criterion. ↩

Von Mises Yield Criterion

[2] In 1913, Richard von Mises, an Austria-Hungary born scientist, proposed a theory for predicting the yielding of ductile materials. The theory states that the yielding occurs when the deviatoric strain energy density (or deviatoric energy for short) reaches a critical value.

For linearly elastic materials, the total strain energy *density* is half of the inner product of the stress and the strain,

$$w = \frac{1}{2} \{\sigma\} \cdot \{\varepsilon\} \quad (6)$$

This total strain energy can be decomposed into two parts: the energy w^p caused by the hydrostatic stress and the energy w^d caused by the deviatoric stress,

$$w = w^p + w^d \quad (7)$$

or

$$w^d = w - w^p \quad (8)$$

The von Mises criterion can be stated as follows: the yielding occurs when

$$w^d \geq w^{yd} \quad (9)$$

where w^{yd} is the deviatoric energy when the material yields in its uniaxial tension test. In the following discussion, we will express w^p and w^d in terms of stresses.

Deviatoric Energy w^{yd}

In a uniaxial tension test, when yielding occurs, the stress state (see 1.4.4[3], page 42) is

$$\begin{Bmatrix} \sigma_y & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{Bmatrix} = \begin{Bmatrix} \sigma_y/3 & 0 & 0 \\ 0 & \sigma_y/3 & 0 \\ 0 & 0 & \sigma_y/3 \end{Bmatrix} + \begin{Bmatrix} 2\sigma_y/3 & 0 & 0 \\ 0 & -\sigma_y/3 & 0 \\ 0 & 0 & -\sigma_y/3 \end{Bmatrix}$$

or, written in a more compact form,

$$\{\sigma^y\} = \{\sigma^{yp}\} + \{\sigma^{yd}\}$$

The first part $\{\sigma^{yp}\}$ is the hydrostatic stress and the second part $\{\sigma^{yd}\}$ is the deviatoric stress. Hooke's law, Eq. 1.2.8(1) (page 31), can be used to obtain the strains. The strains corresponding to the total stress $\{\sigma^y\}$ and the hydrostatic stress $\{\sigma^{yp}\}$ are respectively

$$\{\varepsilon^y\} = \frac{\sigma_y}{E} \begin{Bmatrix} 1 & 0 & 0 \\ 0 & -\nu & 0 \\ 0 & 0 & -\nu \end{Bmatrix} \quad \text{and} \quad \{\varepsilon^{yp}\} = \frac{(1-2\nu)\sigma_y}{3E} \begin{Bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{Bmatrix}$$

Calculated using Eq. (6), the energies corresponding to the total stress and the hydrostatic stress are respectively

$$w^y = \frac{1}{2} \{\sigma^y\} \cdot \{\varepsilon^y\} = \frac{\sigma_y^2}{2E} \quad \text{and} \quad w^{yp} = \frac{1}{2} \{\sigma^{yp}\} \cdot \{\varepsilon^{yp}\} = \frac{(1-2\nu)\sigma_y^2}{6E}$$

By using Eq. (8), the deviatoric energy is

$$w^{yd} = w^y - w^{yp} = \frac{(1+\nu)\sigma_y^2}{3E} \quad (10) \quad \leftarrow$$

Deviatoric Energy w^d

[3] Now, we consider the general 3D stress state (see Eqs. (3-4), page 43). The strains corresponding to the total stress $\{\sigma\}$ and the hydrostatic stress $\{\sigma^p\}$ are, using Eq. 1.2.8(1) (page 31), respectively

$$\{\varepsilon\} = \frac{1}{E} \begin{Bmatrix} \sigma_1 - \nu(\sigma_2 + \sigma_3) & 0 & 0 \\ 0 & \sigma_2 - \nu(\sigma_3 + \sigma_1) & 0 \\ 0 & 0 & \sigma_3 - \nu(\sigma_1 + \sigma_2) \end{Bmatrix} \quad \text{and} \quad \{\varepsilon^p\} = \frac{(1-2\nu)p}{E} \begin{Bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{Bmatrix}$$

Calculated using Eq. (6), the energies corresponding to the total stress $\{\sigma\}$ and the hydrostatic stress $\{\sigma^p\}$ are respectively

$$w = \frac{1}{2} \{\sigma\} \cdot \{\varepsilon\} = \frac{1}{2E} [\sigma_1(\sigma_1 - \nu\sigma_2 - \nu\sigma_3) + \sigma_2(\sigma_2 - \nu\sigma_3 - \nu\sigma_1) + \sigma_3(\sigma_3 - \nu\sigma_1 - \nu\sigma_2)]$$

and

$$w^p = \frac{1}{2} \{\sigma^p\} \cdot \{\varepsilon^p\} = \frac{3(1-2\nu)p^2}{2E}$$

From Eqs. (8, 2), the deviatoric energy is

$$\begin{aligned} w^d &= w - w^p \\ &= \frac{1}{2E} [\sigma_1(\sigma_1 - \nu\sigma_2 - \nu\sigma_3) + \sigma_2(\sigma_2 - \nu\sigma_3 - \nu\sigma_1) + \sigma_3(\sigma_3 - \nu\sigma_1 - \nu\sigma_2)] - \frac{3(1-2\nu)}{2E} \left(\frac{\sigma_1 + \sigma_2 + \sigma_3}{3} \right)^2 \end{aligned} \quad (11)$$

After some manipulations, the above equation can be simplified as

$$w^d = \frac{1+\nu}{6E} [(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2] \quad (12)$$

Derivation of Eq. (12)

$$\begin{aligned} w^d &= \frac{1}{2E} [\sigma_1(\sigma_1 - \nu\sigma_2 - \nu\sigma_3) + \sigma_2(\sigma_2 - \nu\sigma_3 - \nu\sigma_1) + \sigma_3(\sigma_3 - \nu\sigma_1 - \nu\sigma_2)] - \frac{3(1-2\nu)}{2E} \left(\frac{\sigma_1 + \sigma_2 + \sigma_3}{3} \right)^2 \\ &= \frac{1}{2E} [\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - 2\nu(\sigma_1\sigma_2 + \sigma_2\sigma_3 + \sigma_3\sigma_1)] - \frac{1-2\nu}{6E} [\sigma_1^2 + \sigma_2^2 + \sigma_3^2 + 2(\sigma_1\sigma_2 + \sigma_2\sigma_3 + \sigma_3\sigma_1)] \\ &= \frac{1}{6E} [3(\sigma_1^2 + \sigma_2^2 + \sigma_3^2) - 6\nu(\sigma_1\sigma_2 + \sigma_2\sigma_3 + \sigma_3\sigma_1) - (1-2\nu)(\sigma_1^2 + \sigma_2^2 + \sigma_3^2) - (2-4\nu)(\sigma_1\sigma_2 + \sigma_2\sigma_3 + \sigma_3\sigma_1)] \\ &= \frac{1}{6E} [(2+2\nu)(\sigma_1^2 + \sigma_2^2 + \sigma_3^2) - (2+2\nu)(\sigma_1\sigma_2 + \sigma_2\sigma_3 + \sigma_3\sigma_1)] \\ &= \frac{1+\nu}{3E} [\sigma_1^2 + \sigma_2^2 + \sigma_3^2 - \sigma_1\sigma_2 - \sigma_2\sigma_3 - \sigma_3\sigma_1] \\ &= \frac{1+\nu}{6E} [(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2] \end{aligned}$$



Von Mises Stress (Equivalent Stress)

[4] Substituting Eqs. (10, 12) into the von Mises Yield criterion, Eq. (9), we conclude that the material yields when

$$\frac{1+\nu}{6E} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right] \geq \frac{1+\nu}{3E} \sigma_y^2 \quad (13)$$

After simplification, we reach a more concise form

$$\sqrt{\frac{1}{2} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right]} \geq \sigma_y \quad (14)$$

The quantity on the left-hand-side is called *von Mises stress* or *effective stress* and is denoted by σ_e ; in ANSYS, it is also called *equivalent stress*,

$$\sigma_e = \sqrt{\frac{1}{2} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right]} \quad (15)$$

To have more insight into Eq. (14), let's plot Eq. (14) in σ_1 - σ_2 - σ_3 space and consider only equal sign. It will be a cylindrical surface aligned with the axis $\sigma_1 = \sigma_2 = \sigma_3$ and with a radius of $\sqrt{2}\sigma_y$ (see 14.1.4[2], page 530). It is called the *von Mises yield surface*. Condition of Eq. (14) is equivalent to saying that the material fails when the stress state is on or outside the von Mises yield surface. When $\sigma_1 = \sigma_2 = \sigma_3$, the material is under hydrostatic pressure. It is the portion of stress that deviates from the axis $\sigma_1 = \sigma_2 = \sigma_3$ that contributes to the failure of the material.

Equivalent Strain

The *equivalent strain*, or von Mises strain, ε_e is defined by^[Ref 4]

$$\varepsilon_e = \frac{1}{1+\nu'} \sqrt{\frac{1}{2} \left[(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_3 - \varepsilon_1)^2 \right]} \quad (16)$$

Where ν' , the effective Poisson's ratio, defaults to the Poisson's ratio of the material, 0.5, or 0, depending on various applications (for details, see Ref. 4). #

References

1. Haslach, H.W.Jr. and Armstrong, R.W., *Deformable Bodies and Their Material Behavior*, John Wiley & Sons, Inc., 2004; Chapter 9 Plasticity.
2. Cook, R. D. and Young, W. C., *Advanced Mechanics of Materials*, Macmillan, 1985.
3. Beer, F. P., Johnston, E. R. Jr., and DeWolf, J. T., *Mechanics of Materials*, 3rd Ed., McGraw Hill, 2002.
4. All Help>Mechanical APDL>Theory Reference>2.4.1. Combined Strains

Section 1.5

Review

1.5.1 Keywords (Part I)

Choose a letter for each keyword, from the list of descriptions

- | | |
|-------------------------------|---|
| 1. () APDL | 11. () Engineering Data |
| 2. () Boundary Conditions | 12. () Environment Conditions |
| 3. () Brittle Materials | 13. () Failure Criteria of Materials |
| 4. () Buckling | 14. () Finite Element |
| 5. () Degenerated Element | 15. () Finite Element Mesh |
| 6. () Degree of Freedom | 16. () Finite Element Model |
| 7. () DesignModeler | 17. () First-Order Element |
| 8. () Displacement | 18. () Free Boundaries |
| 9. () Ductile Materials | 19. () Governing Equations |
| 10. () Dynamic Simulations | 20. () Isotropic Elasticity |

Answers:

1. (E) 2. (K) 3. (R) 4. (N) 5. (P) 6. (I) 7. (C) 8. (T) 9. (Q) 10. (S)
 11. (A) 12. (L) 13. (O) 14. (B) 15. (G) 16. (H) 17. (J) 18. (M) 19. (D) 20. (F)

List of Descriptions

(A) An application of **Workbench GUI**. It can store material properties. Loads and boundary conditions also can be stored with the application.

(B) A small portion of a problem domain. Its geometry is so simple that the governing equations can be pre-formulated in terms of discrete nodal degrees of freedom.

(C) An application of the Workbench GUI. It is similar to any other feature-based CAD software, except that it is specifically used to create geometric models for use in the ANSYS Workbench simulations.

(D) A set of equations governing the behavior of an engineering system (e.g., a structural system), usually in a form of differential equations.

- (E) ANSYS Parametric Design Language. A set of text commands to drive ANSYS software.
- (F) In **Engineering Data**, this term is used for materials whose stress-strain relation can be described by Hooke's law and characterized by two material parameters: Young's modulus and Poisson's ratio.
- (G) A collection of elements and nodes.
- (H) A finite element mesh plus its environment conditions.
- (I) In the finite element methods, a term used for the discrete nodal values. In Workbench structural simulations, they are nodal displacements.
- (J) Linear polynomials are used as shape functions; also called a linear element, or a lower-order element. Nodes are on the vertices of the element.
- (K) Conditions applied on the boundaries of a finite element mesh.
- (L) Conditions applied on the boundaries or interior of a finite element mesh.
- (M) Boundaries with no boundary conditions specified. ANSYS assumes a zero pressure (stress normal to the surface) on a free boundary.
- (N) When the compression in a structure is large enough such that its lateral stiffness vanishes, the structure becomes unstable.
- (O) For a brittle material, it fails if the maximum principal stress reaches the fracture stress. For a ductile material, Tresca criterion or von Mises criterion may be used. The Tresca criterion states that the material fails if the stress intensity reaches the yield stress. The von Mises criterion states that the material fails if the von Mises stress reaches the yield stress.
- (P) A 3D solid element, hexahedral in its natural shape, may combine some of its nodes to form a triangle-based prism, quadrilateral-based pyramid, or tetrahedron. A 2D solid element, quadrilateral in its natural shape, may combine some of its nodes to form a triangle.
- (Q) The strain is large before it is stretched up to fracture. The fracture is mostly due to a shear failure. There is usually an obvious yield point in its stress-strain curve.
- (R) The strain is small before it is stretched up to fracture. The fracture is mostly due to a tensile failure. There is usually no obvious yield point in its stress-strain curve.
- (S) Structural simulations in which dynamic effects are included.
- (T) In a deformed body, the vector connecting from its initial position to its final position.

1.5.2 Keywords (Part II)

Choose a letter for each keyword from the list of definitions

- | | |
|--|-----------------------------|
| 21. () Linear Simulations | 31. () Stiffness |
| 22. () Mechanical | 32. () Stiffness Matrix |
| 23. () Modal Analysis | 33. () Strain |
| 24. () Node | 34. () Strain State |
| 25. () Nonlinear Simulations | 35. () Stress |
| 26. () Procedure of Finite Element Method | 36. () Stress Intensity |
| 27. () Project Schematic | 37. () Stress State |
| 28. () Principal Stress | 38. () Stress Stiffening |
| 29. () Second-Order Element | 39. () Von Mises Stress |
| 30. () Shape Functions | 40. () Workbench GUI |

Answers:

- 21.(S) 22.(E) 23.(J) 24.(G) 25.(T) 26.(N) 27.(R) 28.(L) 29.(K) 30.(I)
 31.(P) 32.(H) 33.(C) 34.(D) 35.(F) 36.(M) 37.(B) 38.(Q) 39.(O) 40.(A)

List of Definitions

- (A) A gateway to ANSYS applications, including **Project Schematic, Engineering Data, DesignModeler, Mechanical, Design Exploration**, etc.
- (B) A group of values describing the force intensity on the point in all directions of a point in a body. In 3D, three independent directions are needed to complete the description.
- (C) A vector describing the stretch and twist in a direction in a point of a body. The vector can be decomposed into two components: one that is normal to the face and one that is parallel to the face.
- (D) A group of values describing the stretch and twist in all directions of a point in a body. In 3D, three independent directions are needed to complete the description.
- (E) An application of **Workbench GUI**. It performs structural and mechanical simulations, including meshing, setting up environment conditions, solving finite element models, and viewing results.
- (F) The force per unit area acting on the boundary surfaces of an infinitesimally small body centered at a point of a body.
- (G) Entities connecting elements. The elements share the same degrees of freedom values on the entities.

(H) The matrix that describes the linear relation between displacement vector and the force vector. Physical meaning of the i^{th} column is the forces required on each of the DOFs to maintain a unit displacement on the i^{th} DOF and zero displacements on the other DOFs.

(I) In finite element methods, they are used as interpolating functions to calculate continuous displacement fields from discrete nodal displacements. Linear and quadratic polynomials are commonly used as the interpolating functions.

(J) Also called free vibration analysis. It is a special case of dynamic analysis, in which the structure is free of external forces. The results of the analysis include natural frequencies and their corresponding vibration modes.

(K) Quadratic polynomials are used as shape functions, also called a higher-order element. Nodes are on the vertices as well as on the middle of the edges.

(L) At a point in a body, different directions (faces) have different stress values. There exist directions in which the normal components are in their extremities and the shear components vanish; the directions and the corresponding normal stresses are called the principal directions and the principal stresses respectively. In 3D, there are 3 principal stresses; they are denoted as, in order starting from the largest, σ_1 , σ_2 , and σ_3 respectively. It is often used in a criterion for a brittle material: when the maximum principal stress is larger than the fracture strength, the material fails.

(M) Defined as the difference between the maximum principal stress and the minimum principal stress. It equals to twice the maximum shear stress. It is used in Tresca failure criterion for a ductile material: when it is larger than the yield strength, it is equivalent to saying that the shear stress is larger than the yielding shear strength and, thus, the material fails.

(N) (a) Calculate stiffness matrix for each element according to the element's geometry and its material properties. (b) Add up the element matrices to form a global stiffness matrix. The force vector is also calculated, according to the loading conditions. (c) Eq. 1.3.1(1) (page 35) is solved for the nodal displacements. (d) For each element, the displacement fields are calculated according to Eq. 1.3.2(2) (page 36); the strain fields are calculated according to Eq. 1.2.7(1) (page 31); the stress fields are calculated according to Eq. 1.2.8(3) (page 32).

(O) It is used in a failure criterion for a ductile material: when it is larger than the yield strength, it is equivalent to saying that the distortion strain energy density is larger than the yielding distortion strain energy density and, thus, the material fails.

(P) The forces required to deform the structure.

(Q) The phenomenon that, when a structure member is subject to a tensile stress, its lateral stiffness increases with the increase of the tensile stress.

(R) An application of **Workbench GUI**. Its function is to lay out simulation systems and their data flows.

(S) The responses of a system are linearly proportional to the loads.

(T) The responses of a system are not linearly proportional to the loads.

Section 1.6

Appendix: An Unofficial History of ANSYS

Shen-Yeh Chen, Ph. D.

About the Author

Dr. Shen-Yeh Chen earned his BS degree in civil engineering from National Chung Hsing University, Taiwan, in 1990, and his MS and Ph.D. degrees in structural mechanics from Arizona State University, USA, in December of 1997.

He began to work for Honeywell Engines and Systems in Phoenix, Arizona, in February of 1998. Shen-Yeh was a heavy user of ANSYS and LS-DYNA during his service at Honeywell. Soon he became a cross-department and cross-campus expert of ANSYS and LS-DYNA at Honeywell, and was very active on the internet (the XANSYS group) in the ANSYS internal users community. Dr. Chen published several popular ANSYS macros and articles during that time, and developed many in-house codes for Honeywell, including an optimizer to couple with ANSYS and LS-DYNA.

In August of 2002, Dr. Chen took a VP position offered from CADMEN, an ANSYS distributor in Taiwan. In 2005, Dr. Chen established his own company, FEA-Opt Technology (www.FEA-Optimization.com). About 80% of the revenue for FEA-Opt Technology is related to ANSYS and LS-DYNA. In 2006, he released a general purpose design optimization software, SmartDO. Today FEA-Opt Technology is well-known as a very successful CAE consulting firm in Taiwan, and its SmartDO is gaining more and more market share since 2006.

Dr. John Swanson holds B.S. and M.S. degrees in mechanical engineering from Cornell University. He holds a Ph.D. in applied mechanics from the University of Pittsburgh, obtained in night school with Westinghouse support.

In 1963, Dr. John Swanson worked at Westinghouse Astronuclear Labs in Pittsburgh, responsible for stress analysis of the components in NERVA nuclear reactor rockets. He used computer codes to model and predict transient stresses and displacements of the reactor system due to thermal and pressure loads. Swanson continued to develop 3D analysis, plate bending, nonlinear analysis for plasticity and creep, and transient dynamic analysis, in the next several years, using a finite element heat conduction program that was developed by Wilson at Aerojet. The old Westinghouse codes included a 2D/axisymmetric one also, possibly called FEATS (according to Kohnke). John wanted to combine these codes to remove the duplication, like equation solvers and some postprocessing.

Swanson believed an integrated, general-purpose FEA code could be used to do complex calculations that engineers typically did manually, such as heat transfer analysis. It would save money and time for Westinghouse and other companies.

Westinghouse didn't support the idea, and Swanson left the company in 1969. Before he left, he made sure that all code work had been sent to COSMIC, so that he could pick it up again from the outside.

Swanson Analysis Systems, Inc. was incorporated in the middle of 1970 at Swanson's home. The offices were part of Swanson's home (there was no garage) in Pittsburgh. At the same time, Westinghouse realized that they needed John, so they hired him as a consultant. John said sure, but with the proviso that whatever he put into STASYS, the Westinghouse code, he could also put into ANSYS. Westinghouse had no trouble with this, as they just wanted to solve their problems. So this consulting kept bread on the table for the Swansons, and at the same time brought forth further improvements to ANSYS.

He developed his program using a keypunch and a time-shared mainframe at U.S. Steel. The first version of ANSYS was coded by the end of 1970, and the ANSYS program was first released soon after that. Westinghouse was the first customer, running as a data center. The data center was at the Telecommunications Center on Parkway East, on the east side of Pittsburgh. According to Dr. Swanson, the name ANSYS was used because the copyright lawyers assured Swanson that ANSYS was just a name, and did not stand for anything. This is understandable, because during that period all programs were "written" on punch card. When installing the program on the customer's computer, it meant carrying a relatively big case of punch cards to the customer's place, and feeding them into the machine.

Dr. Peter Kohnke met John Swanson first about early 1971. Swanson offered Peter a job in the fall of '71, but Peter did not accept. At that time Peter was a brand new father, and Westinghouse looked a lot more secure than SASI. Peter told John he was interested and would accept in the spring of '72, but then he had hired Gabe DeSalvo, and did not have the resources to hire Peter also. But John was finally able to hire Peter in the fall. Dr. Peter Kohnke's start date was 1/1/73.

When Peter started work, he asked what John wanted him to do. John told Peter that he developed code, did technical support, wrote manuals, gave seminars and did systems work so that the program would run on a variety of systems. John said he needed relief, so Peter should pick one or two of them. Ultimately, Peter did all of them, except systems work.

In around 1970, users ran ANSYS 2.x on a CDC 6600 machine over the Cybernet timesharing network. At the time, only fixed format input was available. The users would work up the input listing off-line, key it onto a tape cassette, log on, submit the run about quitting time for the best computer rates and stop by the CDC data center next morning to find out what went wrong. In 1973, ANSYS ran on three kinds of hardware: CDC, Univac, and IBM. And around 1973 the USS mainframe that they used to develop code was the US Steel CDC 6500.

The first minicomputer that ANSYS ran on was a MODCOMP 4 (or IV?). VAX came later. Being a small company, everyone did everything. When a "mini" computer was delivered, everyone helped wrestle it off of the truck. When printout paper was delivered, everyone helped unload the boxes.

In 1975, MITS began to build and sell the first PC ever in human history, the Altair. That, of course, did not have anything to do with ANSYS yet. The so-called PC was just a few switches and lights on the front board, and input had to be done in a binary fashion (no keyboard and monitor, of course). What was worse was that you had to assemble it by yourself, and it usually didn't work. Although Altair was rather popular, nobody really knew what to do with this machine. One former customer said that the most popular activity on Altair was to figure out what to do with this machine. At the same time, Microsoft built the BASIC language for Altair.

In 1977, Apple I was born.

In around 1979, Revision 3.0, ANSYS ran on a VAX 11-780 minicomputer. ANSYS evolved from fixed format input to purely command line driven and monochrome (green) on a Tektronix 4010 or 4014 vector graphics monitor. For a decent size model, the hidden lines plots could take 20-30 minutes. All of the nodes and elements were created separately without the benefit of importing CAD geometry. NGEN, EGEN, RPnnn, were used extensively. There was a geometry preprocessor, PREP7.

In 1980, we had Apple II.

In around 1980, John Swanson bought a Radio Shack TRS-80 machine, and planned to build a commercial version on it. However, later John returned the machine because Radio Shack left out (a socket for) a floating point processor. John decided that Finite Element Analysis probably should utilize a floating point processor, so he got his money back for that one.

Also around 1980, Rev 4 on a VAX 11-780 system was great, according to some old users. The chasm between batch and interactive running pretty much disappeared and file management was a very easy thing. No more element hard coding, the post processing got hugely better and you could mix batch and interactive running as you saw fit. Big dynamic transient runs or substructuring over night, post-processing and plotting next morning. Emag capabilities were first introduced at Rev 4.1.

Also in 1980, Microsoft signed a contract with IBM to provide the OS (PC DOS) for its up coming PC. This OS, however, was not created by Microsoft. Microsoft bought it from an engineer for 50K USD; and it was named the QDOS - the Quick and Dirty Operation System.

In 1981, IBM PC was born. This computer was created using the off the shelf technology, and an open architecture. The original reasons were to push the product to the market ASAP, so that IBM could catch up with the PC market. However, the BIOS was proprietary. Later Compaq reverse-engineered the BIOS and created a fully IBM PC compatible BIOS. This ignited the PC cloning market and war. The booming of the PC market directly changed the meaning of computing. The price of a PC dropped 30% in one month. And, it was the booming of the cloned IBM PC that really brought money into Microsoft.

In 1984, the revolutionary Macintosh was born. Macintosh was far more advanced then the IBM PC family at that time. The concept of GUI in the OS level and WYSIWYG was not possible on IBM PC until almost one decade later. However, the market of Macintosh did not pick up very soon, which caused Steve Jobs to leave Apple computer.

However, later the sales of Macintosh began to take off, which proved that Steve Jobs' vision had all been right. Macintosh saved Apple, and was directly responsible for the Apple phenomena.

A PC version of ANSYS was also available at around version 4.0 in about 1984. It was running on an Intel 286, with interactive command line input and limited graphics on the screens, like elements and nodes. No Motif GUI yet. In the first release of ANSYS on PCs, preprocessing, solution and post processing were performed in separate programs.

"Design Optimization" was introduced on Rev 4.2 (1985). This is also the release at which "Macro length is no longer limited to 400 characters."

FLOTRAN started as a graduate (PhD) project by Rita J. Schnipke at the University of Virginia circa 1986. After grad school Rita started (or helped start) Compuflo which was later sold to ANSYS in 1992. Rita later started her own shop which is in Charlottesville, VA, called Blue Ridge Numerics. They make CFDesign, a finite element based CFD code (www.cfdesign.com).

In 1988 at an ANSYS conference in California, IBM was there pushing their first unix machine, the "RT". It was slow. They asked Dr. Swanson if he would make a comment on it. He said "RT" must stand for Real Turkey.

SASI first started working with Compuflo (FLOTRAN) in 1989. In ANSYS Rev 5.0 and FLOTRAN V2.1A, SASI had what they called a "seamless interface" between the two programs (1993). FLOTRAN was "fully integrated" into ANSYS in Rev 5.1 (1994).

In 1993, Version 5.0 was released. And later the version 5.1 had a Motif GUI, which would have few changes to its layout up to 6.0.

Swanson Analysis Systems, Inc., was sold to TA Associates in 1994. The new company name, ANSYS, Inc., was announced at AUTOFACT '94 in Detroit.

According to many different people in the old SASI, John Swanson treated the people there pretty well. In contrast to the old "sandwich" jokes, John never passed up a chance to go to a restaurant. Indeed, for many years, John invited the entire staff to a restaurant regularly for the staff meetings.

At one time, Kohnke told Swanson that he ran the company like a benevolent dictatorship, and later Swanson told Kohnke that he liked that characterization.

Many people have said that John Swanson had an amazing overall understanding as well as detailed knowledge of the ANSYS code. Kohnke told a small story in an email to the author: "Sometime in the late '70s, a bug came my way. I wrestled with it for maybe half a day without making real progress. Then I went to John's office to ask him if he had any ideas. After I explained the bug, John thought about it for about 3 seconds (literally!) and said: 'Didn't you make a change in XXX about 6 months ago that would have a bearing on this?' In a nutshell, he was correct and I was then able to resolve the bug! John's knowledge and understanding of the code was always amazing to me."

In 1995, Windows 95 was published. Windows 95 was an important milestone for Microsoft. It bridged between the old DOS OS and the new NT technology. The birth of Windows 95 finally made it more and more acceptable for the engineering community to use the PC as a heavy duty calculation machine like a workstation.

In 1996, ANSYS 5.3 was published, with support for LS-DYNA. The feature of ANSYS/LS-DYNA in ANSYS 5.3 was still in the beginning stage.

On June 20, 1996, ANSYS Inc. common stock began trading on Nasdaq under ANSS after being a privately held company for 26 years. The IPO generated more than \$41 million.

In 1998, ANSYS began to ship ANSYS/ed to university labs and paper reviewers. One of the copies arrived at the Structures Lab of Civil Engineering Department in Arizona State University, and that was the first time the author knew about ANSYS.

In the same year, on ANSYS's Annual report, it said, "John is retiring from his direct role at ANSYS Inc., but will continue his association as a key consultant, mentoring all of us for many years to come."

On August 31, 2000, ANSYS acquired ICEM CFD.

January 2001, ANSYS announced the release of CADfix (International TechneGroup Incorporated) for ANSYS version 5.6.2 and 5.7. CADfix was to address the issue of importing CAD models into ANSYS with automatic geometric data repair.

In November 2001, ANSYS acquired CADOE S.A., an independent software vendor that specializes in the CAD/CAE market. In the same month, ANSYS announced a strategic OEM partnership with SAS LLC, a provider of NASTRAN simulation software and services. The alliance was focused on the joint development of a new NASTRAN computer-aided engineering solution that will be distributed exclusively by ANSYS Inc.

In November 2001, ANSYS announced AI*Environment. AI*Environment combines ICEM CFD Engineering's pre- and post-processor technologies.

In December 2001, ANSYS 6.0 was released. In this version, the Sparse solver was greatly improved. Efficient and reliable large scale model analysis (say, 1M DOF) finally became practical. The graphics screen of ANSYS was also painted blue in 6.0, which turned out to be a great disappointment to a lot of users.

In April of 2002, ANSYS 6.1 was released. The familiar Motif GUI was replaced by a Tcl/tk developed interface. It runs on 64-bit Intel Itanium architecture with Windows XP.

In February 26, 2003, ANSYS acquired CFX. ANSYS also announced that the functionality of Flotran would be "capped" at 8.1. That is, there will be no more development of Flotran after 8.1. Except for the Multiphysics platform, Flotran will be replaced by CFX.

In March 2004, ANSYS announces ParaMesh 2.3.

In May 2004, ANSYS 8.1 and CFX 5.7 was released. As previously mentioned, there will be no more revision of Flotran after 8.1.

In June 2004, ICEM CFD 5.0 was released.

On Jan 5, 2005, ANSYS announced that it acquired Century Dynamics. Century Dynamics' main product, AUTODYN, includes computational structural dynamics finite element solvers (FE), finite volume solvers for fluid dynamics (CFD), mesh-free particle solvers for high velocity, large deformation and fragmentation problems (SPH), and multi-solver coupling for multiphysics solutions including coupling between FE, CFD and SPH methods.

On February 16, 2006, ANSYS signed a definitive agreement to acquire Fluent.

As of March of 2012, Dr. Peter Kohnke is still working for ANSYS, but has cut back to 4 days a week. He turned 71 in December of 2012.

Dr. John Swanson is officially retired and living in The Villages, in Florida. But he still programs for ANSYS under contract on a varying schedule, on projects such as High performance mesh interpolation, Symmetric Multi Processing, 64 bit conversions, and APDL enhancements.

John is currently on the Board of Trustees of the University of Pittsburgh and the ASME Foundation and served two six-year terms as a Trustee of Washington and Jefferson College. He is a member of the Engineering College Council at Cornell University. His support of colleges and universities includes the donation of research laboratories to the Engineering Schools at Cornell, the University of Pittsburgh and (with Janet, his wife) the Veterinary School at Cornell. He gave the naming gift for the John A. Swanson Science Center at Washington and Jefferson College. The John A. Swanson School of Engineering at the University of Pittsburgh is named in his honor. Swanson recently invested in Applied Quantum Technology (AQT), a California startup company with the objective of reducing the cost of PV Solar Power by another factor of two. He serves on the AQT Board of Directors.

What is a data center?

I actually heard the word "data center" for the first time from Dr. Kohnke. For my age, I am more familiar with Workstation and PC and Xbox and PSP and so on. I used VMS when I was in college. And I remember later soon the whole room was occupied by PC and Macs. So I asked Dr. Kohnke what is a data center anyway? And this is what he told me.

Long ago (in the computer age), computers were relatively rare (and expensive). So, if you invested in one, you wanted to get the maximum use of it. You would run it day and night. This is unlike our PCs now (like our cars), which frankly just sit there most of the time, waiting for our command. Before it was very much a shared and continually used resource.

So, ANSYS got its start in that environment. Users at Westinghouse wanted to run ANSYS, so Westinghouse made a contract with SASI, where SASI would supply the code and support, and Westinghouse would pay to SASI so much per unit of time royalty for the time that the computer was actually being used to run ANSYS. As a result, many users were using the same machine (one after the other--parallel did not exist then). This was called a "data center."

The next step was external users. Knowing that even as a large company you might not need your computer full time, you would have your sales people out there selling time on it, often only at night, to external users.

Other companies sprang up that had no user base of their own, but only a computer(s), sales people, and external customers. The better ones also offered their own very good technical support. These were also called data centers.

And of course the whole concept of a data center disappeared as computers got cheaper and faster.

The "lease" jumped to the concept that the clock was not important; this "new" contract had SASI being paid so much per month, regardless of actual usage. This is closer to how things are now.

Acknowledgement

The author wants to acknowledge the help from many engineers and scientists in the XANSYS internet group. Some of the former employees of ANSYS also contributed greatly to this article, and many of them prefer not to be named. I also received emails from different people, and I usually tried to verify before I used them. Although I am trying to keep all the statements as accurate as possible, I really cannot guarantee the correctness of any information in this article.

Many of us, including the author in the XANSYS group, especially want to thank Dr. John Swanson, who invented ANSYS, and changed the life of many engineers forever in certain ways.

I have lived in the States for a total of 10 years. I lived there, was educated there, married there, had my son there, bought my first new car there, and had my first house there. It totally changed my life, my thinking (and my head) and everything. And I have to say most of my financial support was built on the existence of ANSYS and another program: LS-DYNA.

This article is also available on the web site www.FEA-Optimization.com. Anyone is welcome to distribute this article any way he or she wants, as long as the original article remains unchanged. Comments and suggestions should be forwarded to the authors directly. I will be glad to update this file continuously.

Shen-Yeh Chen

Chapter 2

Sketching

A 3D geometry can be viewed as a series of adding/removing material of simple solid bodies. Each solid body is often created by first drawing a 2D sketch, called a profile, and then extruding/revolving/sweeping the profile to generate the 3D solid body.

Purpose of This Chapter

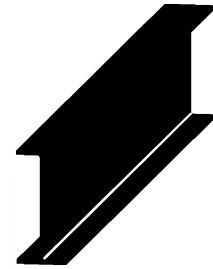
This chapter provides exercises for the students so that they know how to draw 2D sketches using an ANSYS Workbench's geometry editor, DesignModeler. The profiles of several mechanical parts will be sketched in this chapter, and each sketch is then used to generate a mechanical part using a 3D modeling tool such as **Extrude** or **Revolve**. The use of these 3D modeling tools is trivial so that we may focus on 2D sketching techniques. More sophisticated use of 3D modeling tools will be introduced in Chapter 4.

About Each Section

Each mechanical part will be completed in a section. Section 2.1 sketches a cross section of W16x50; the cross section is then extruded to become a 3D beam. Section 2.2 sketches a triangular plate; the sketch is then extruded to become a 3D plate. Section 2.3 does not provide a hands-on case; rather, it overviews the sketching tools in a systematic way, attempting to complement what was missed in the first two sections. Sections 2.4, 2.5, and 2.6 provide three additional exercises, in which we purposely leave out some steps for the students to figure out the details themselves.

Section 2.1

W16x50 Beam



2.1.1 About the W16x50 Beam

[1] In this section, we will create a W16x50 steel beam (see [2-5]).
The beam has a length of 10 ft. ✓

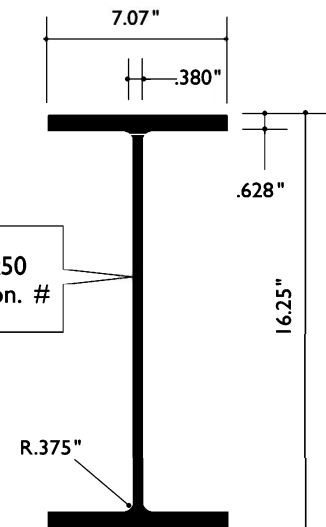
[2] Wide-flange I-shape
section. →

[3] Nominal
depth 16 in. →

[4] Weight 50 lb/ft. →

W16x50

[5] W16x50
cross section. #



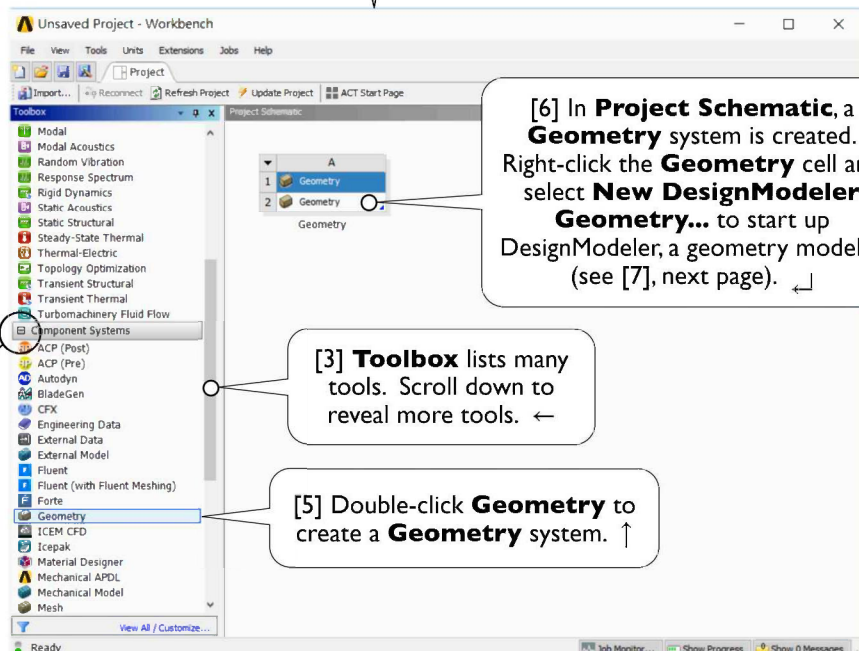
2.1.2 Start Up DesignModeler

[2] **Workbench GUI**
appears. ↓



[1] Launch
Workbench. ✓

[4] Click the plus sign (+) to
expand **Component
Systems**; the plus sign
becomes a minus sign (-). →



[6] In **Project Schematic**, a
Geometry system is created.
Right-click the **Geometry** cell and
select **New DesignModeler
Geometry...** to start up
DesignModeler, a geometry modeler
(see [7], next page). ↵

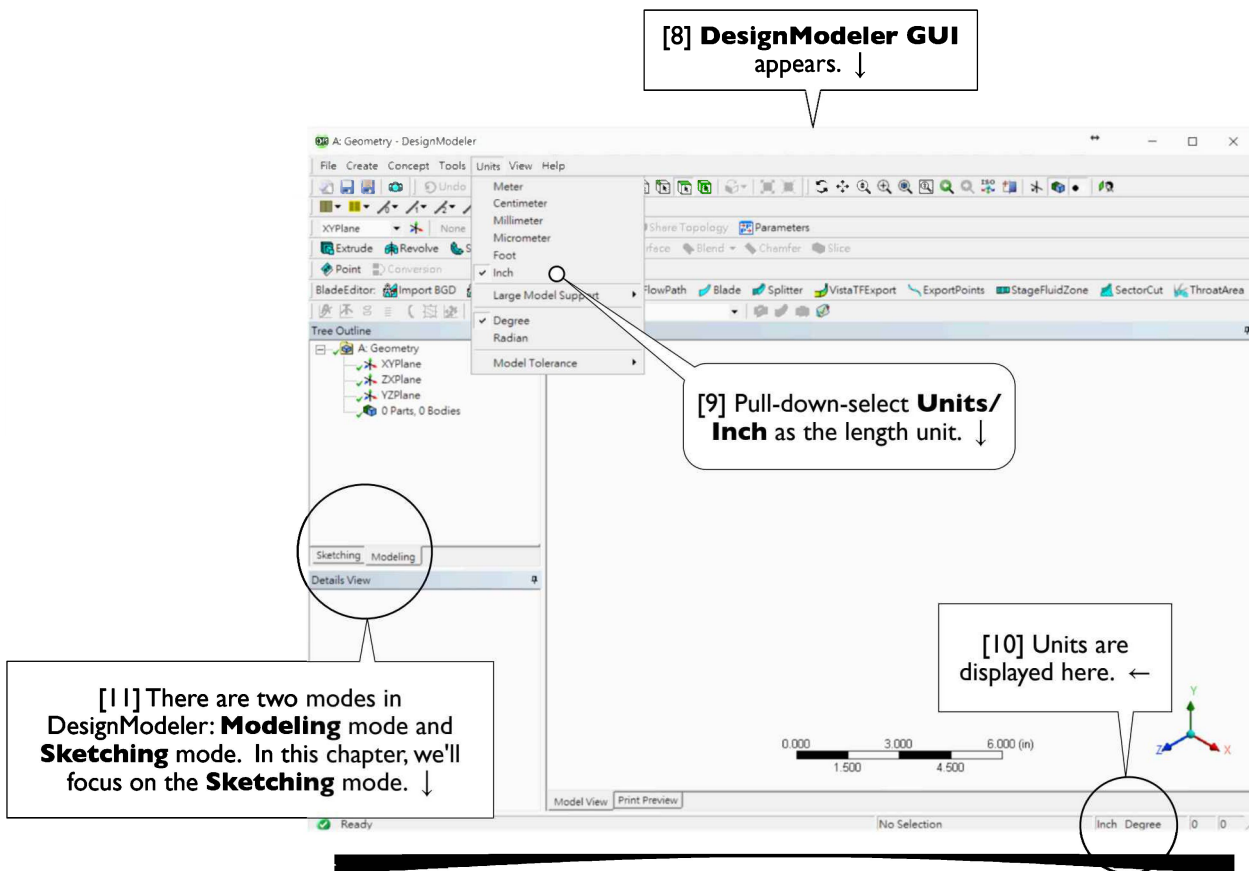
[3] **Toolbox** lists many
tools. Scroll down to
reveal more tools. ←

[5] Double-click **Geometry** to
create a **Geometry** system. ↑

DesignModeler vs. SpaceClaim

[7] As mentioned in 1.1.4[2], page 13, Workbench provides two geometric modelers: **DesignModeler** and **SpaceClaim**. Until ANSYS 15, **DesignModeler** was the only modeler provided by Workbench. For simple and small models, **DesignModeler** serves well enough; but for complicated and large models, the engineers often create a geometric model using a CAD software such as SOLIDWORKS, PTC Creo, Autodesk Inventor, etc., and then import the model to Workbench. In ANSYS 16 and 17, **SpaceClaim** was included in Workbench as an alternative modeler, and **DesignModeler** remained as the default modeler. Starting from ANSYS 18, **SpaceClaim** becomes the default modeler, and **DesignModeler** serves as an alternative modeler.

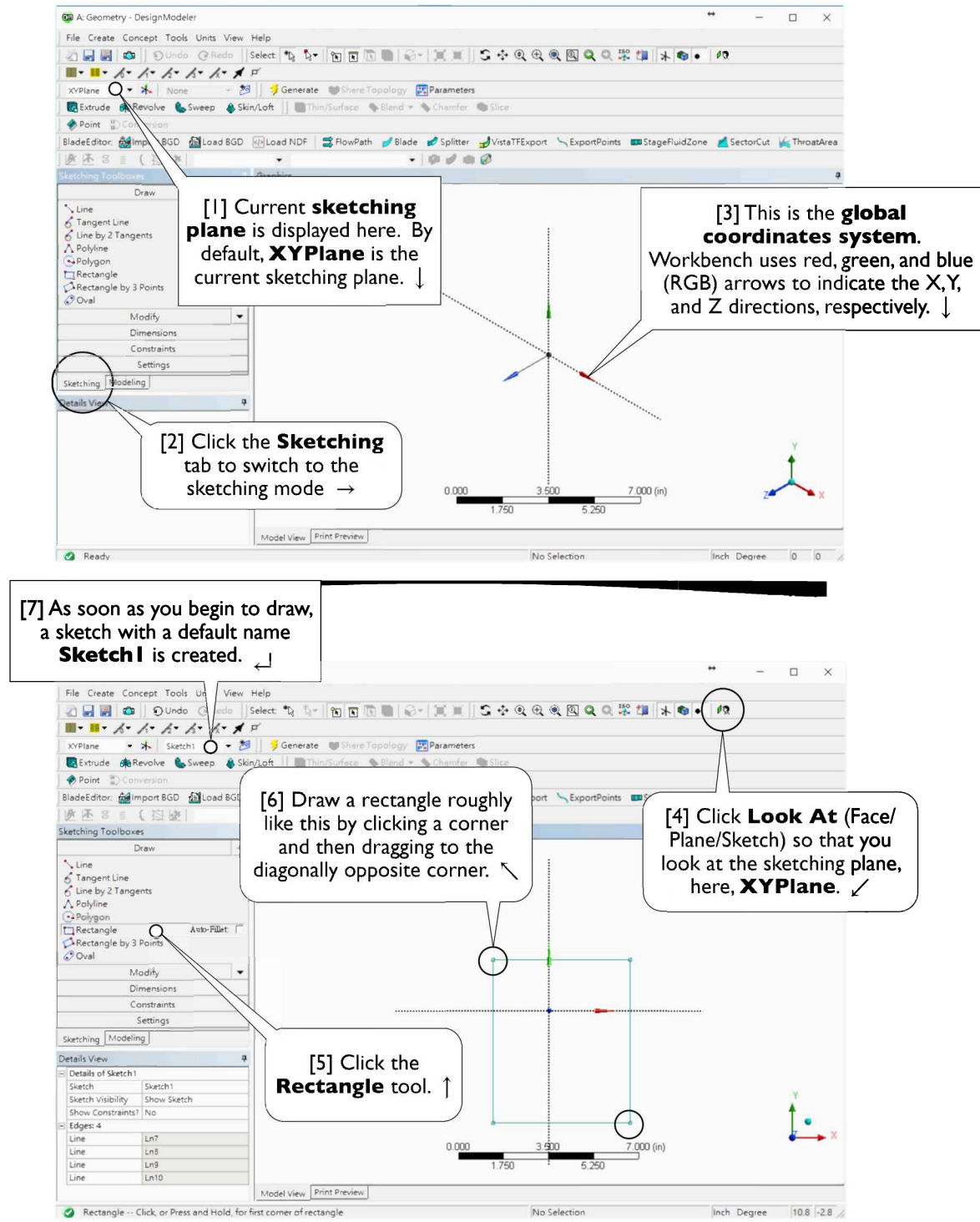
In this book, since all the geometric models are simple and small, we will use **DesignModeler** to create the geometric models. Remember, the focus of this book is the finite element simulations, not the geometric modeling. ✓

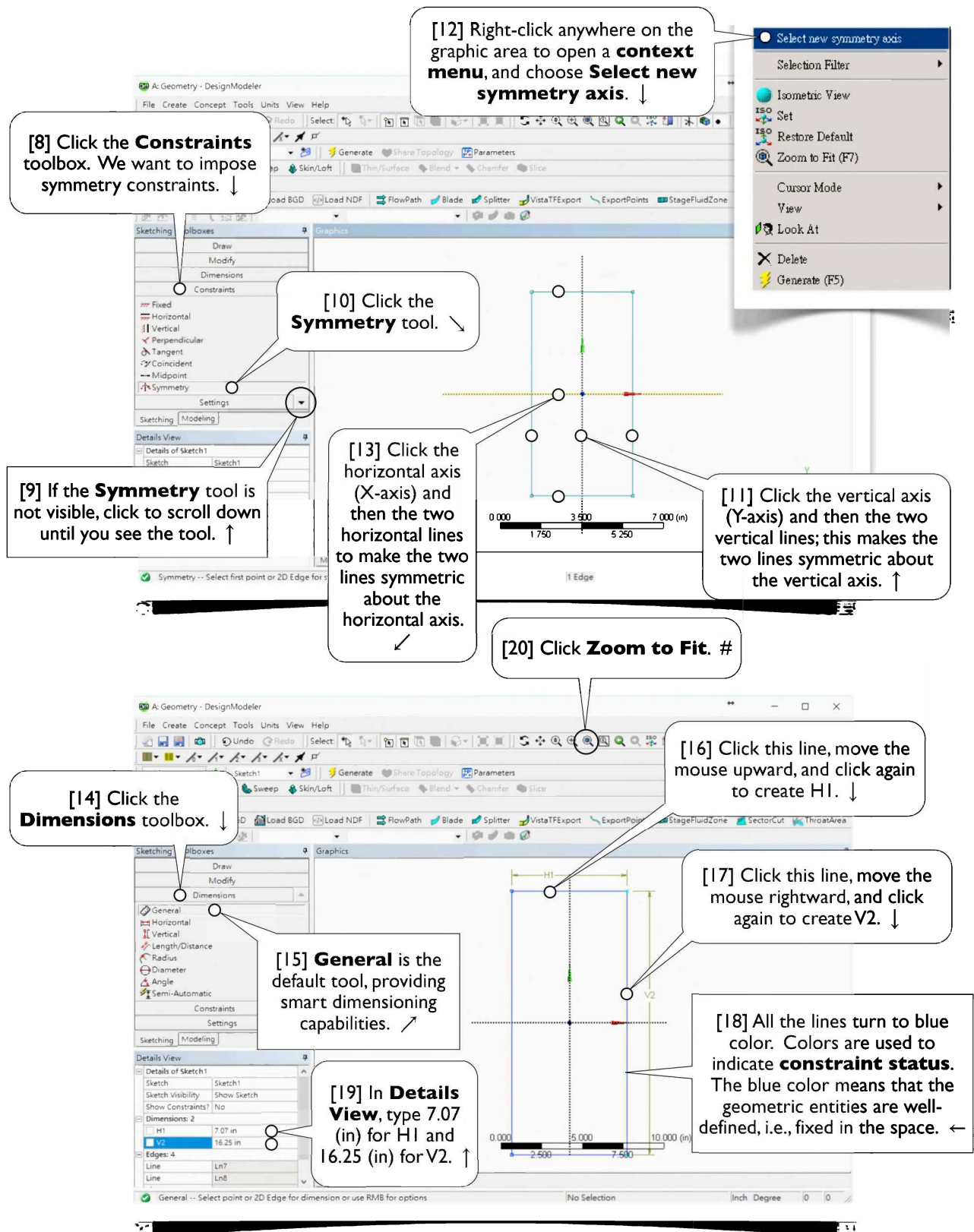


About Textboxes

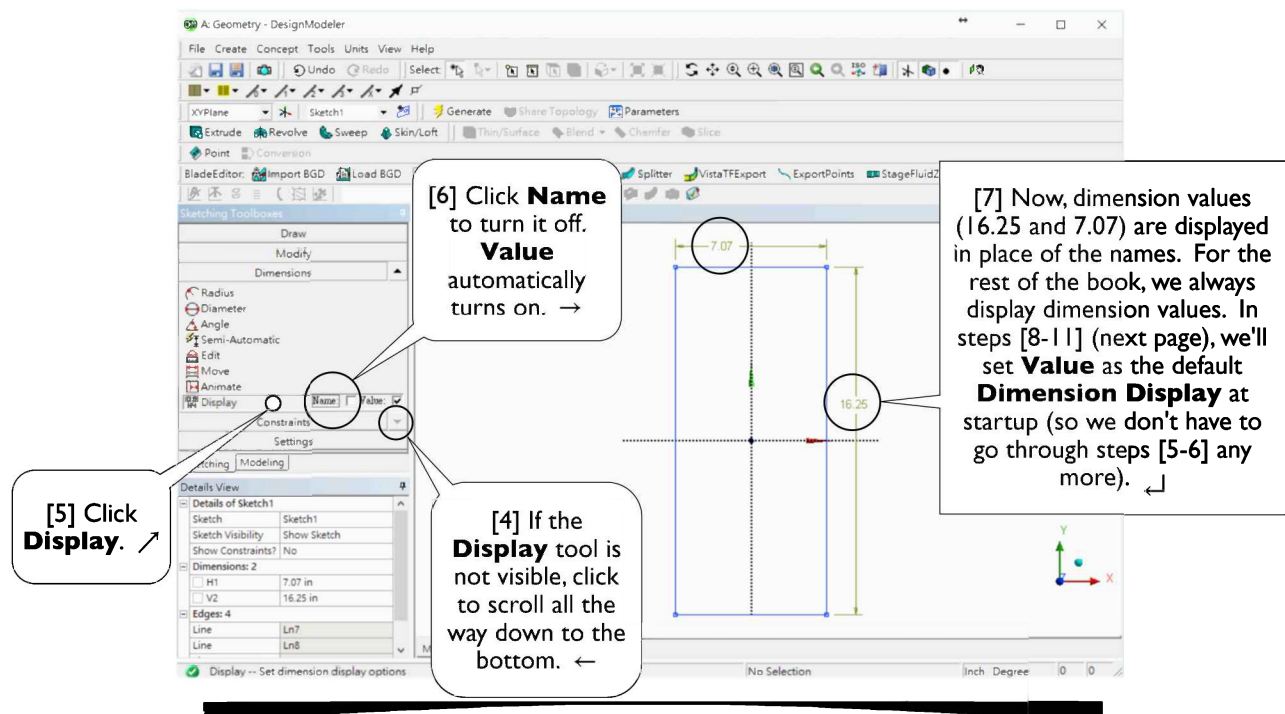
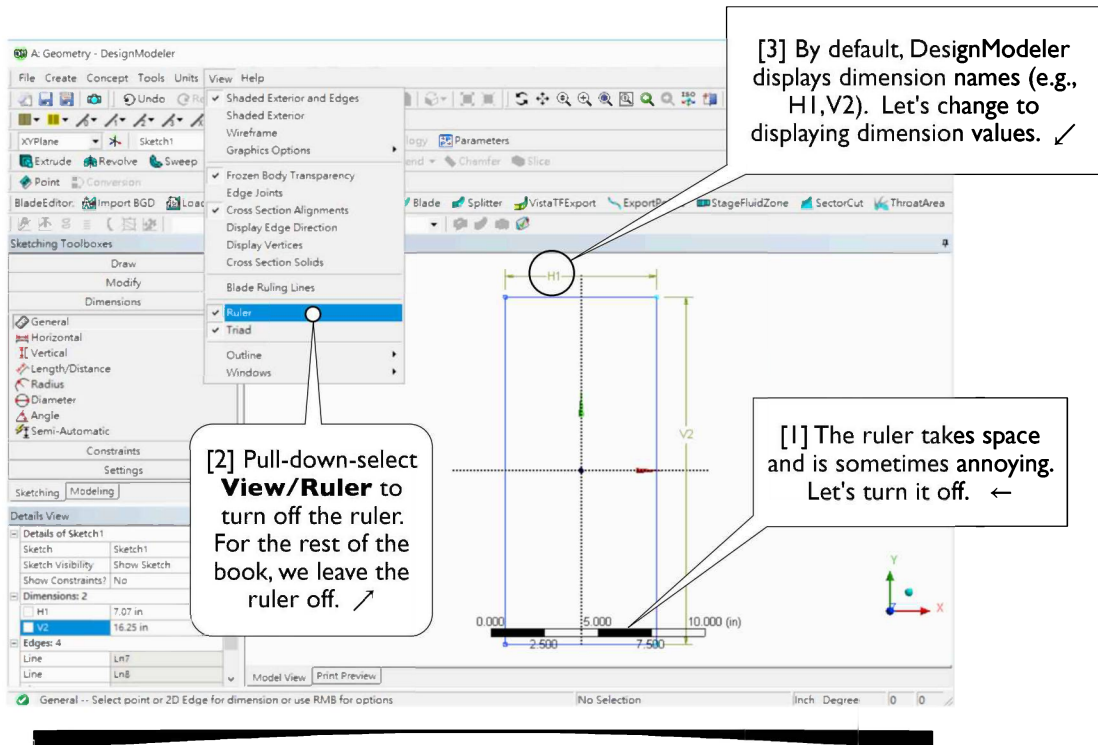
[12] In this book, a round-cornered textbox (e.g., [1, 3-6, 9]) is used to indicate that mouse or keyboard ACTIONS are needed in that step. A sharp-cornered textbox (e.g., [2, 7-8, 10-11]) is used for COMMENTS only; no mouse or keyboard actions are needed in that step. #

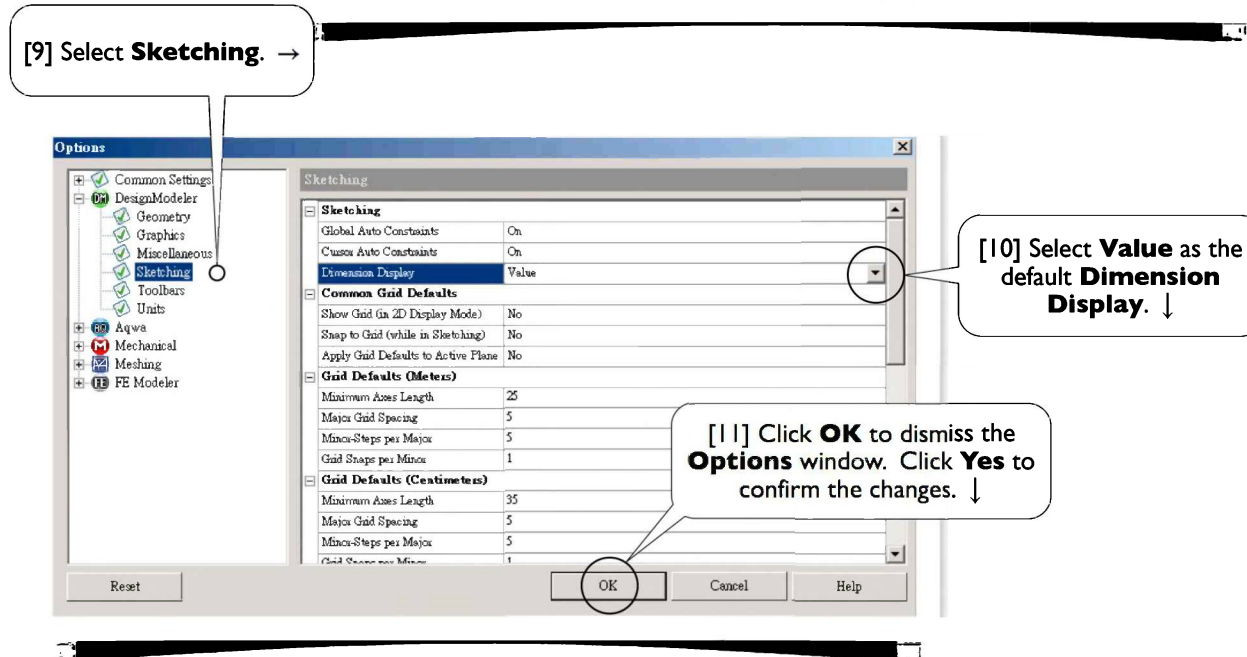
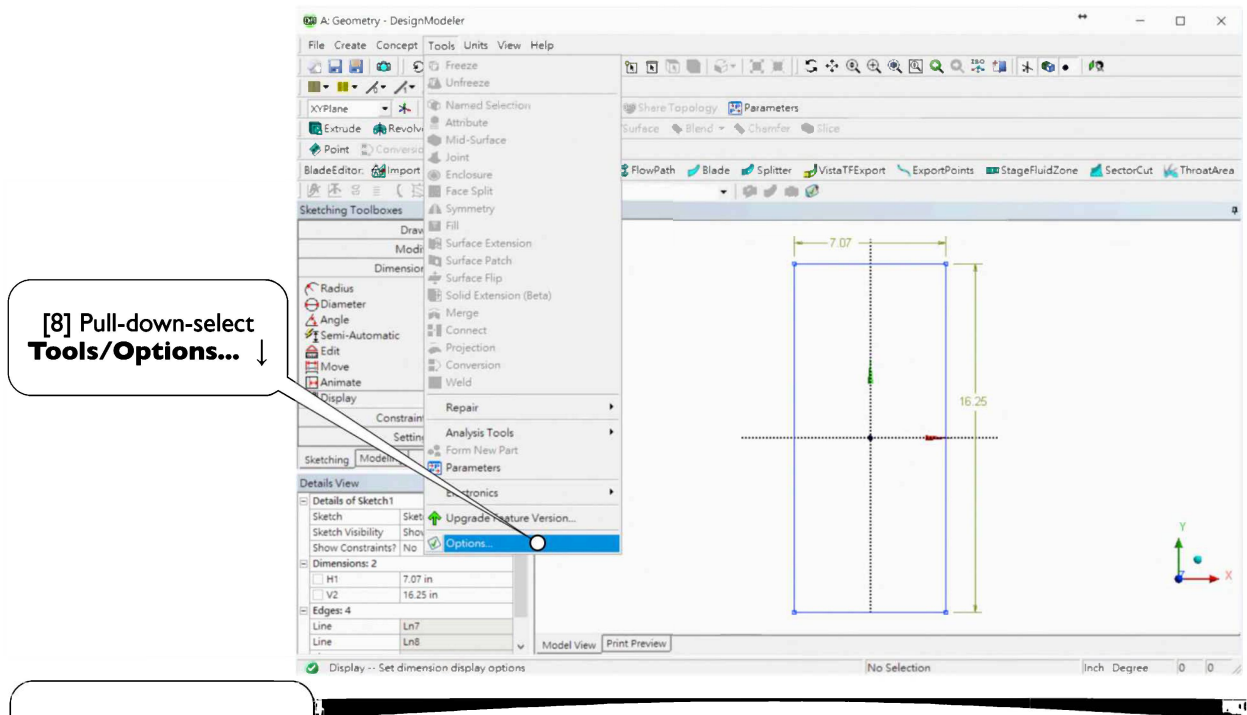
2.1.3 Draw a Rectangle on **XYPlane**





2.1.4 Set Up Sketching Options

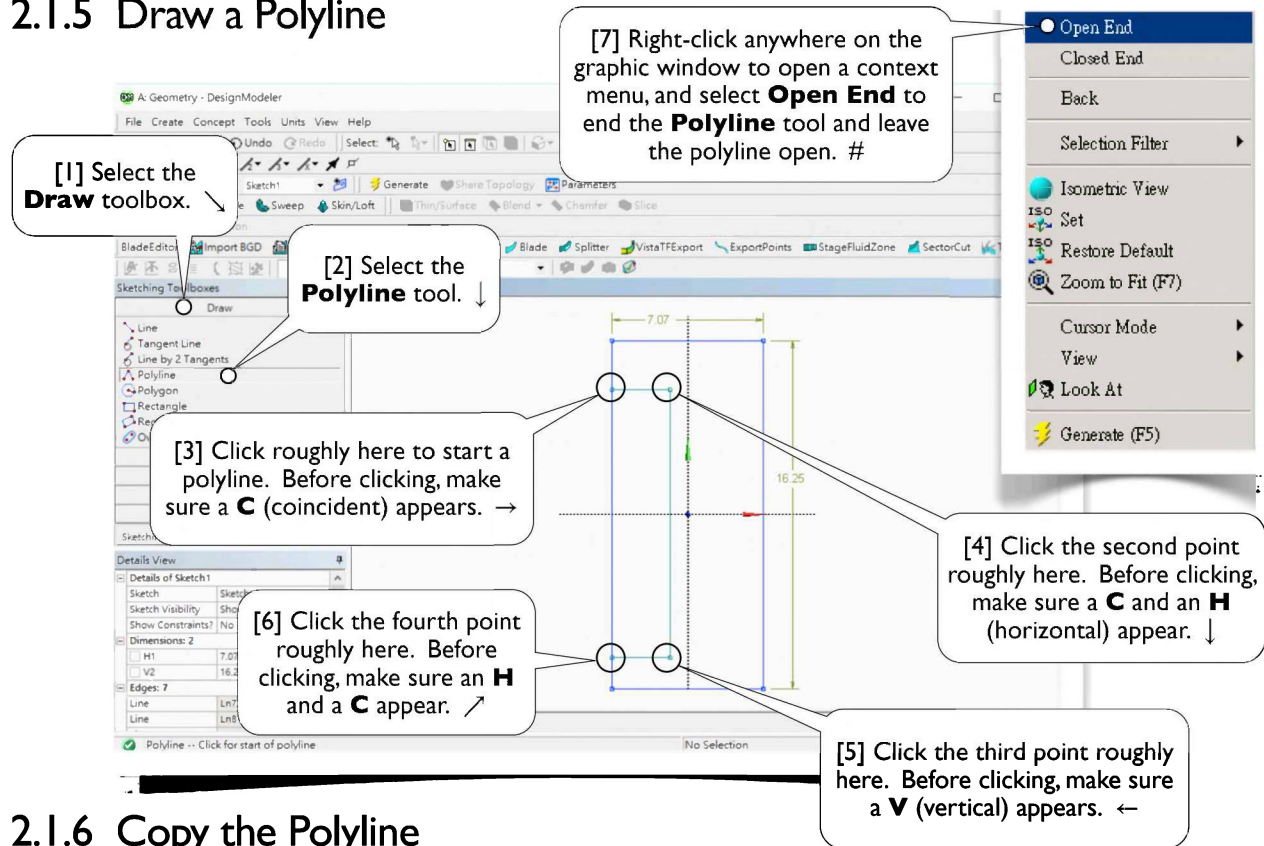




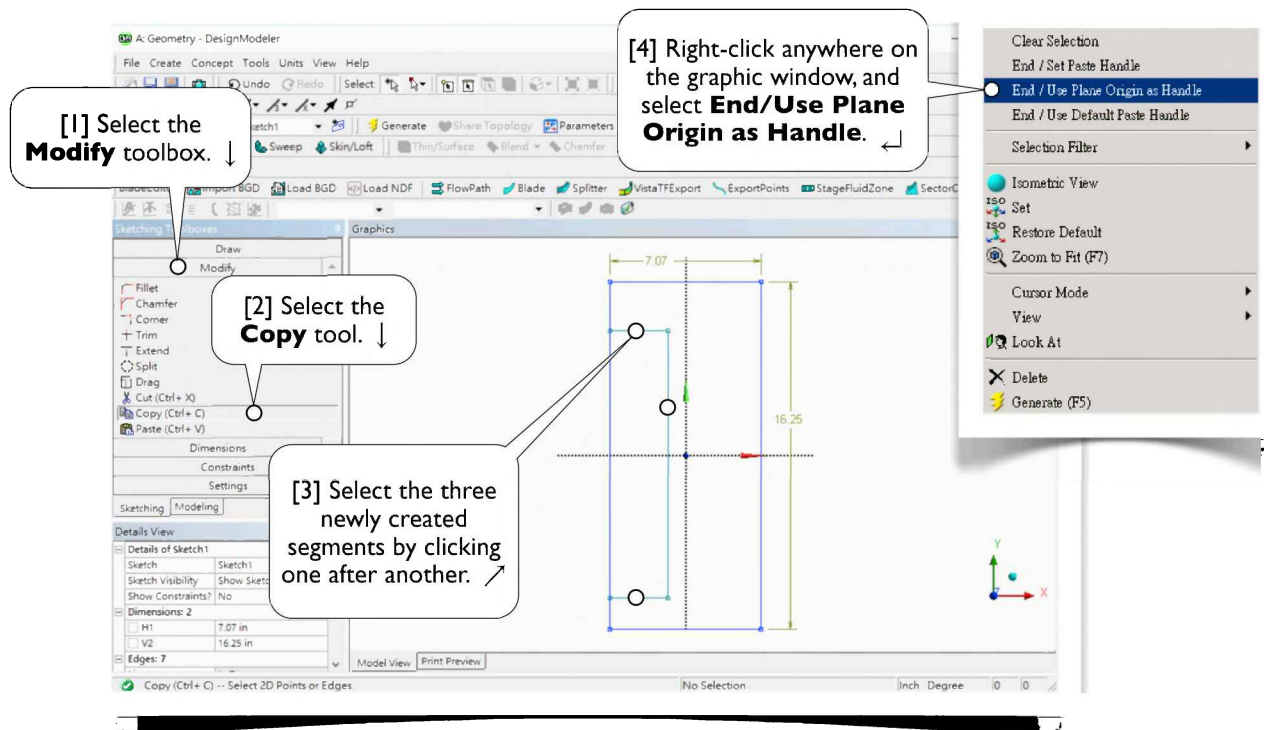
Background Color of the Graphic Area

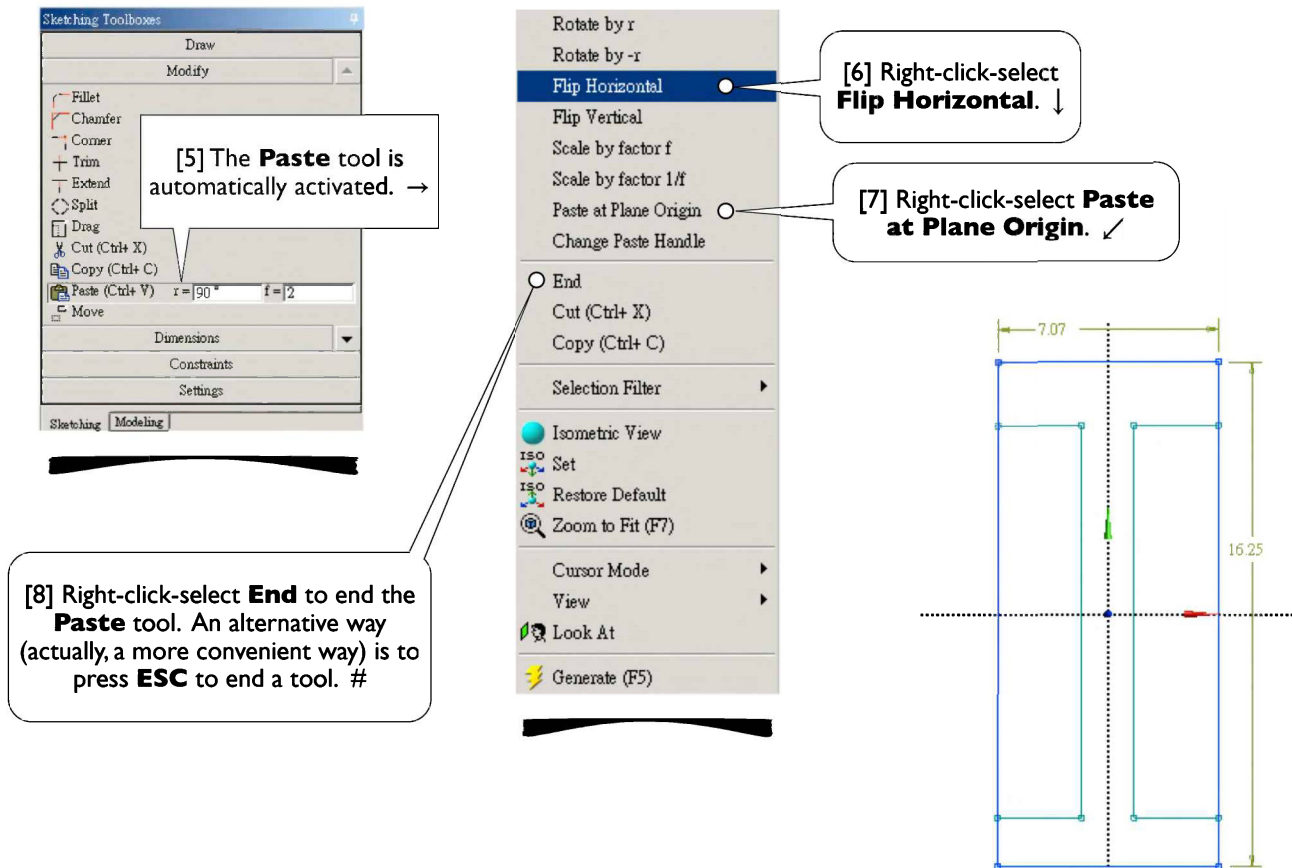
[12] In this book, for better readability, the background color of the graphic area is always shown in white. To set up the background color, pull-down-select **Tools/Options** in **Workbench GUI** (2.1.2[2], page 57; not **DesignModeler GUI**, 2.1.2[8], page 58) and select **Appearance**. #

2.1.5 Draw a Polyline



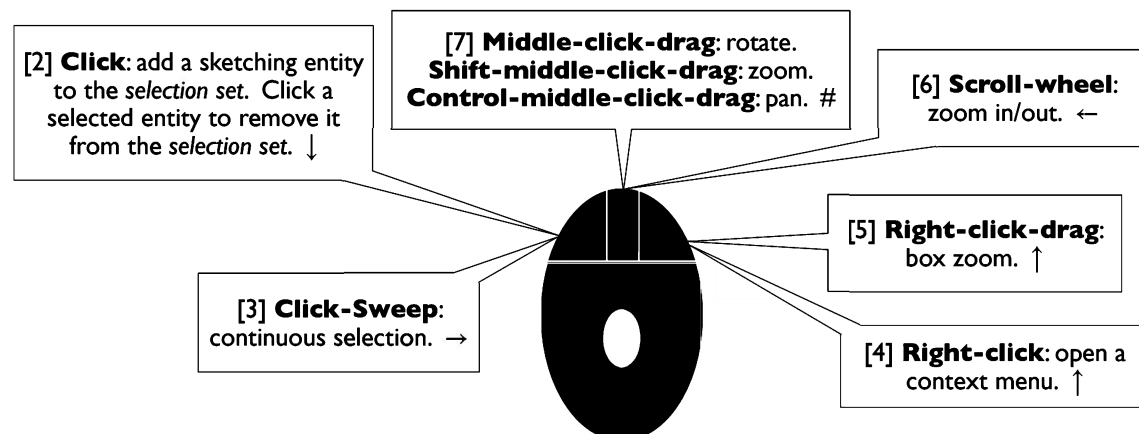
2.1.6 Copy the Polyline



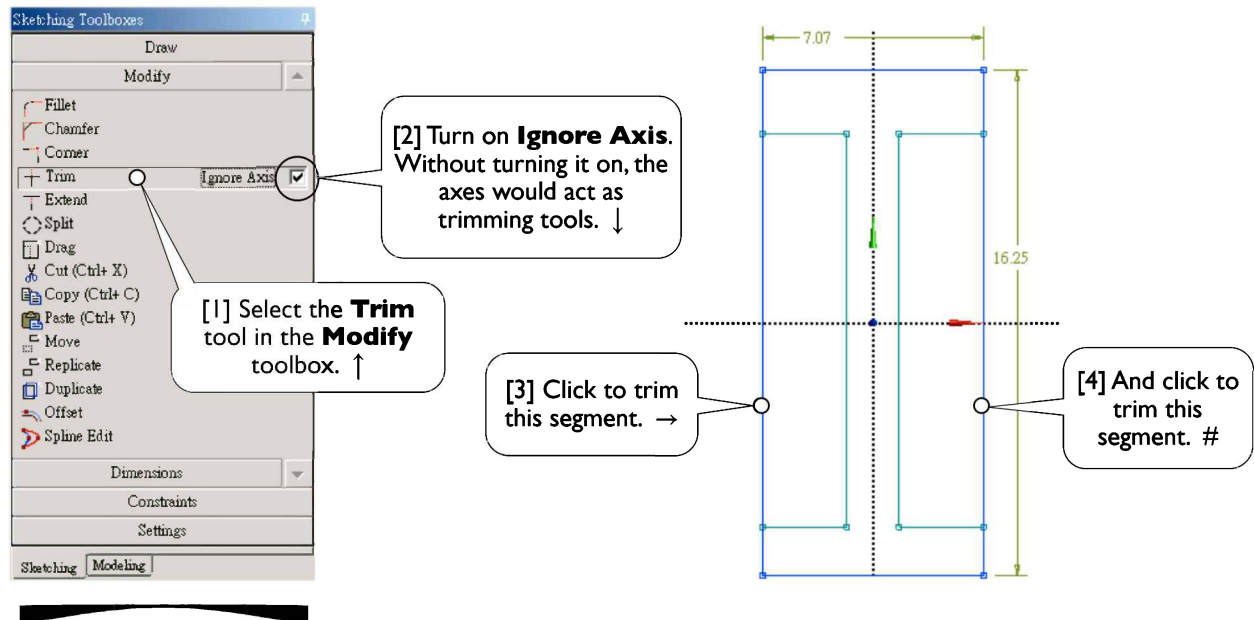


2.1.7 Basic Mouse Operations in Sketching Mode

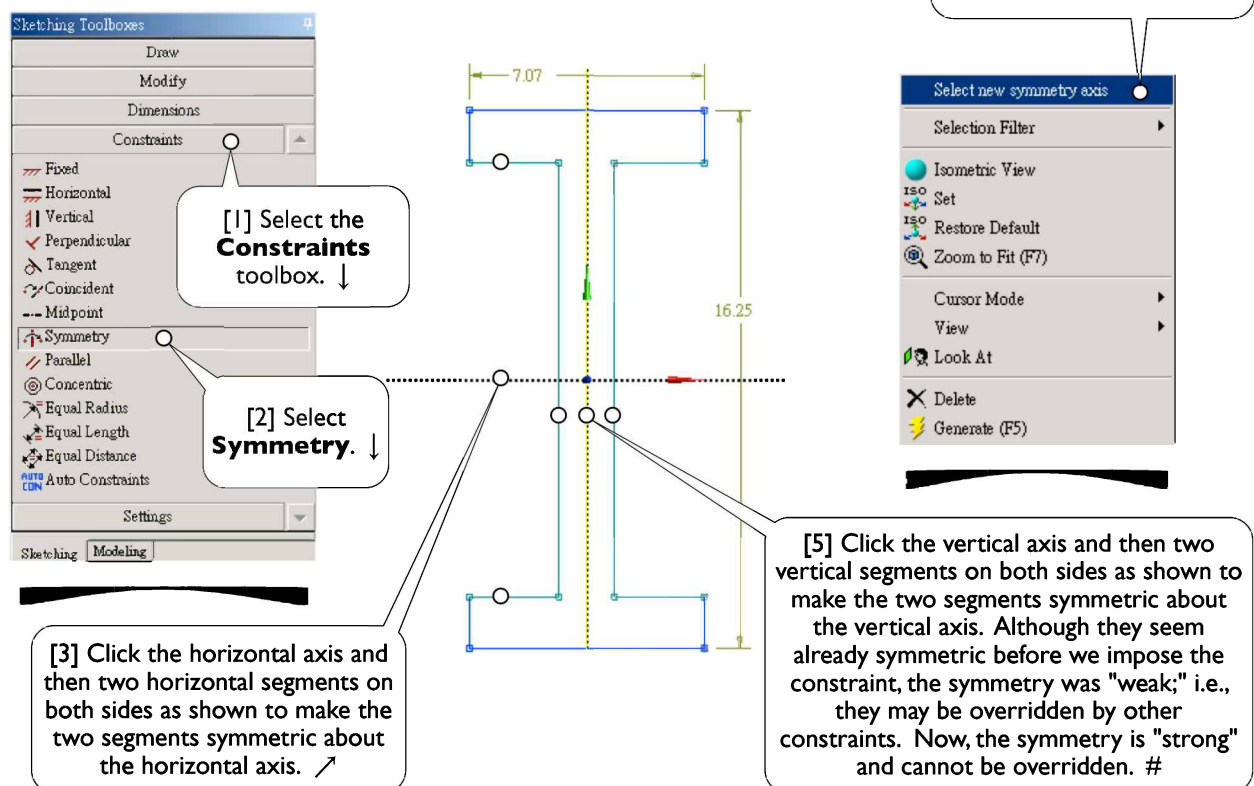
[1] Now, try these basic mouse operations in the sketching mode [2-7]. Press **ESC** to deselect all entities. After trying any of [5-7], click **Zoom to Fit** (2.1.3[20], page 60) or **Look At** (2.1.3[4], page 59) to display a fitting view. ✓



2.1.8 Trim Away Unwanted Segments



2.1.9 Impose Symmetry Constraints



2.1.10 Specify Dimensions

[1] Select the **Dimensions** toolbox. ↓

[2] **General** is the default tool. ↓

[5] Select **Horizontal** (to specify a horizontal dimension). ↓

[3] Click this vertical segment and move leftward to create a dimension (in the details view [4], the corresponding name is V3). ↓

[7] In **Details View**, type 0.38 (in) for H4. ↘

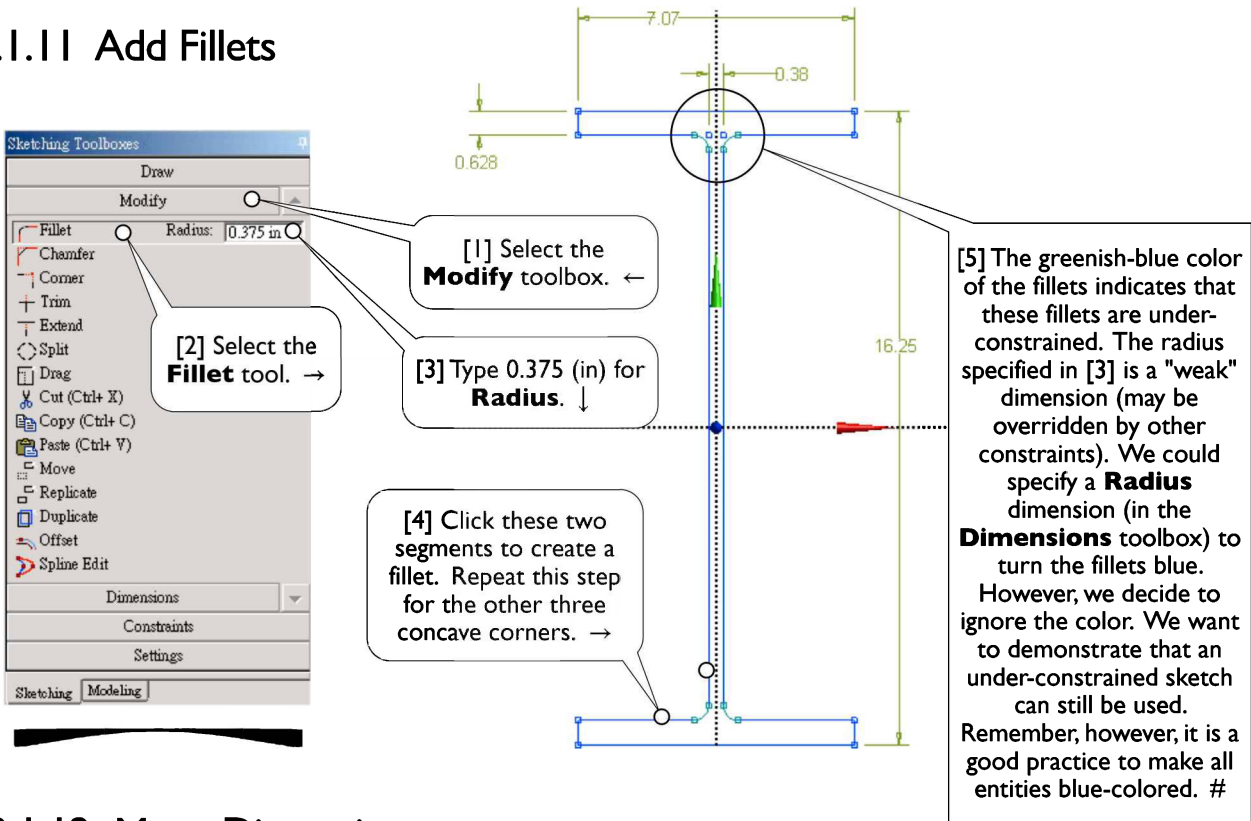
[4] In **Details View**, type 0.628 (in) for V3. ↑

[6] Click these two vertical segments one after the other and move upward to create a horizontal dimension (H4). ↙

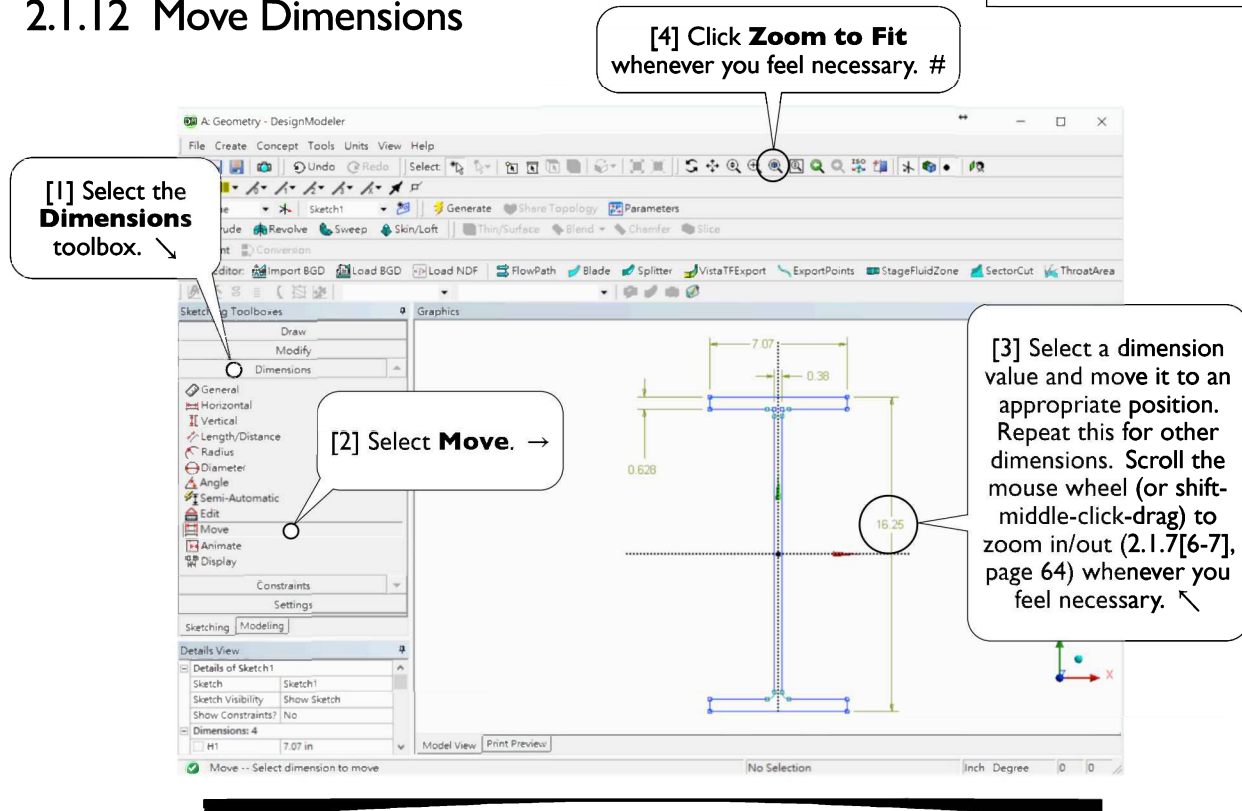
[8] All the sketching entities are blue-colored, meaning that they are well-defined, i.e., fixed in the space. #

Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 4	
H1	7.07 in
H4	0.38 in
V2	16.25 in
V3	0.628 in

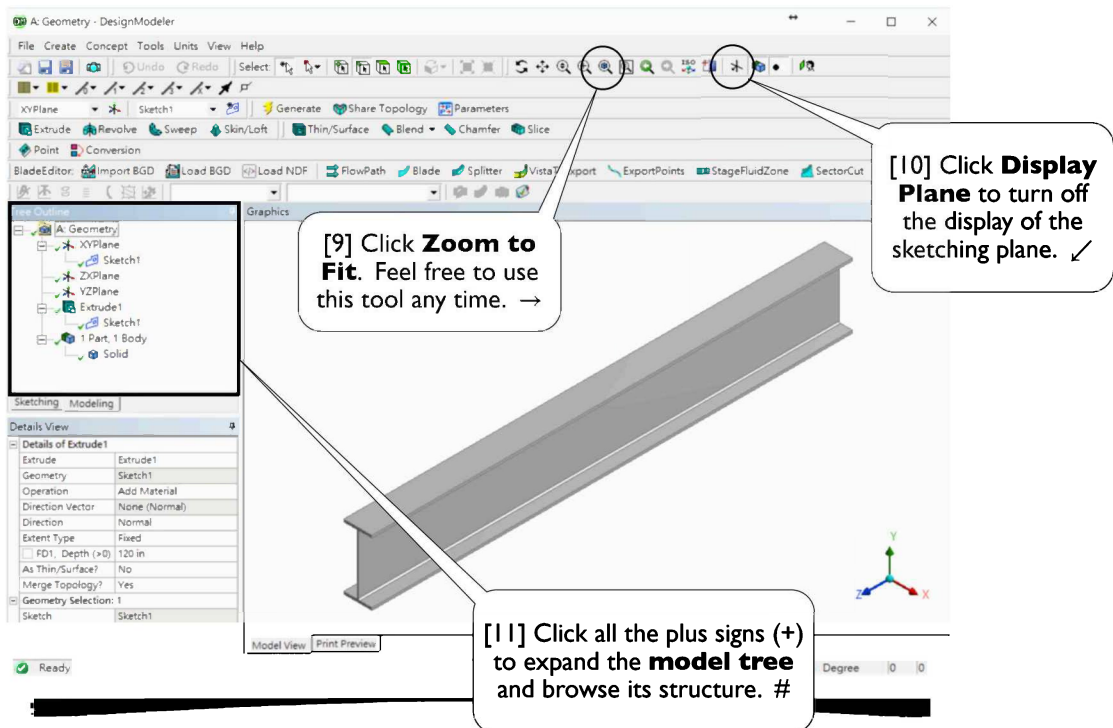
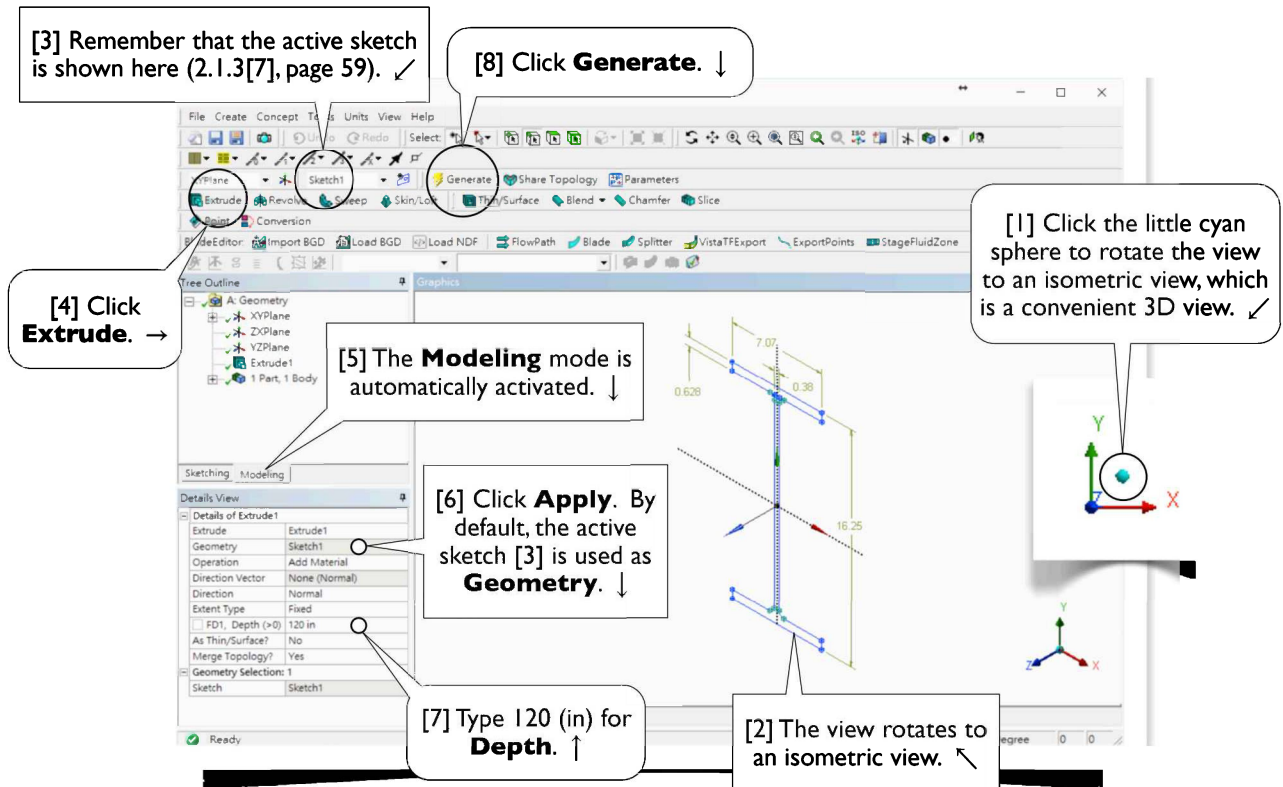
2.1.11 Add Fillets



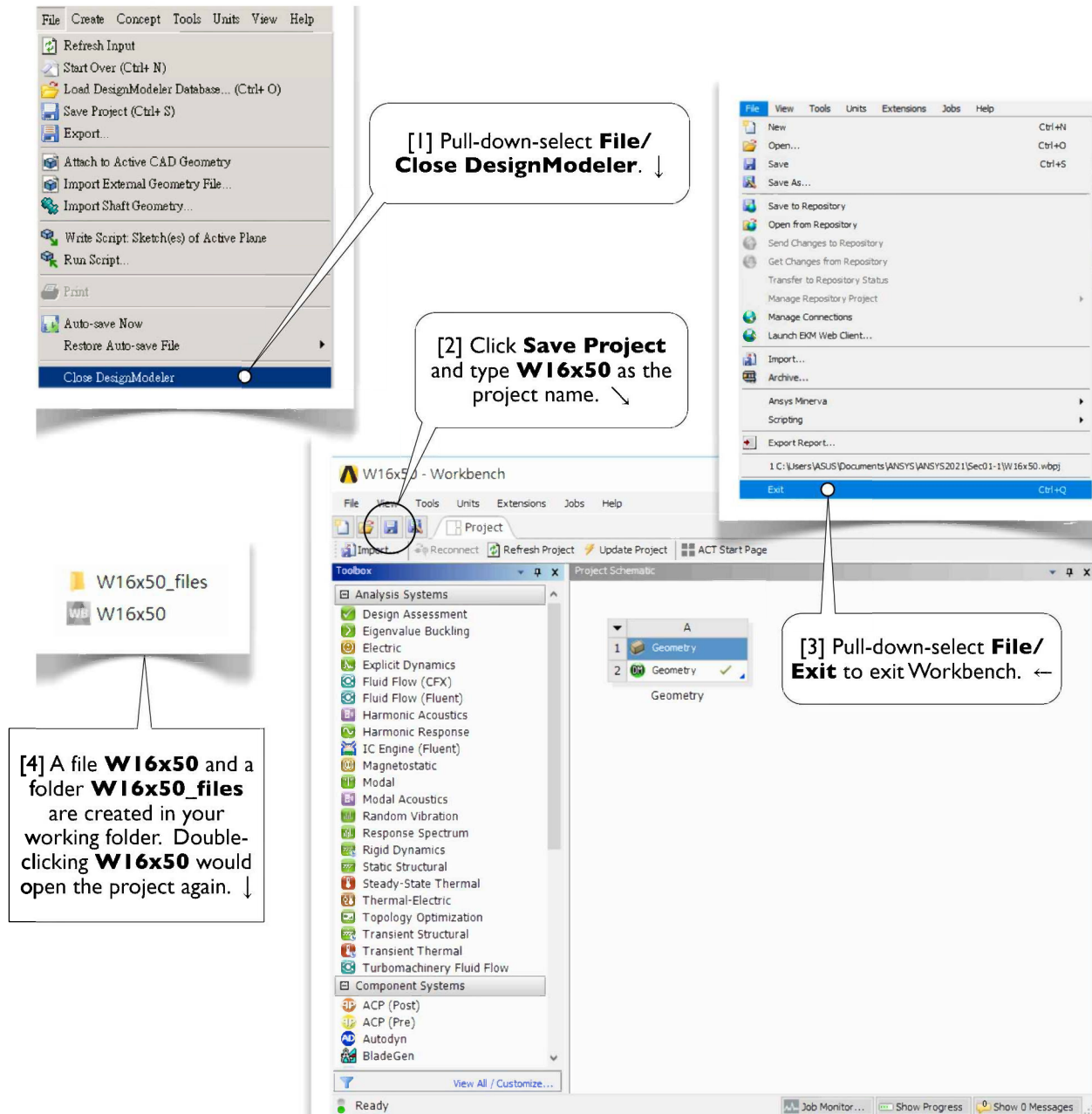
2.1.12 Move Dimensions



2.1.13 Generate a 3D Solid Body



2.1.14 Save Project and Exit Workbench



Supporting Files

[5] To download the finished project files or view the demo videos, please visit SDC Publications' website or the author's webpage. See Access Code and Author's Webpage in **Preface** (page 7) for details. #

Section 2.2

Triangular Plate



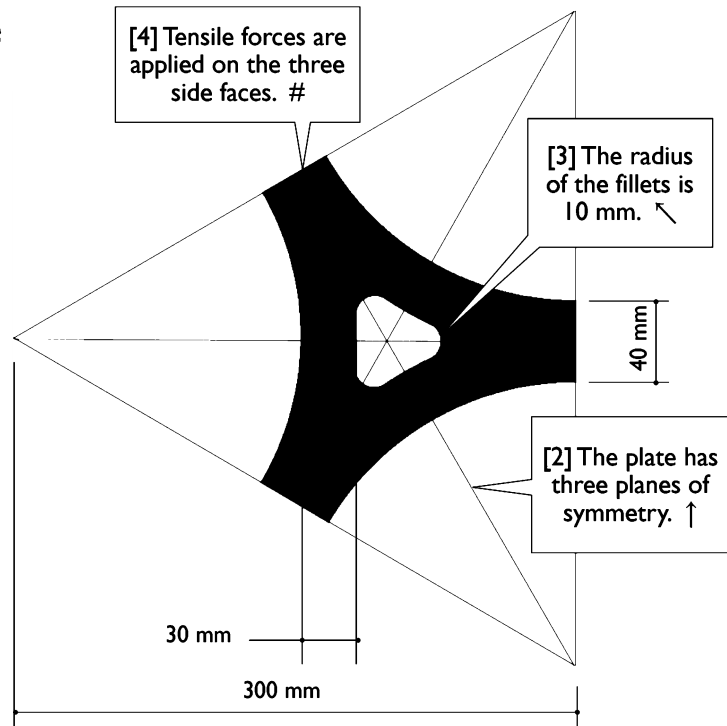
2.2.1 About the Triangular Plate

[1] A triangular plate [2-3], with a thickness of 10 mm, is used to withstand tensile forces on its three side faces [4].

In this section, we'll sketch a profile of the plate on **XYPlane** and then extrude the profile a thickness of 10 mm along Z-axis to generate a 3D solid body.

In Section 3.1, we will use this sketch again to generate a 2D solid model, which is then used for a static structural simulation to assess the stress under the tensile forces.

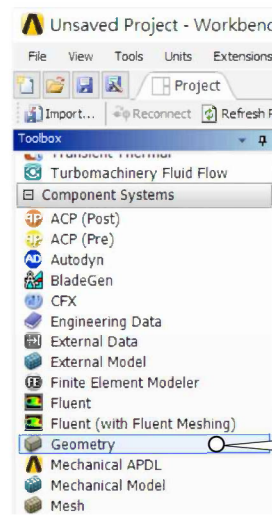
The 2D solid model will be used again in Section 8.2 to demonstrate a design optimization procedure. →



2.2.2 Start up DesignModeler

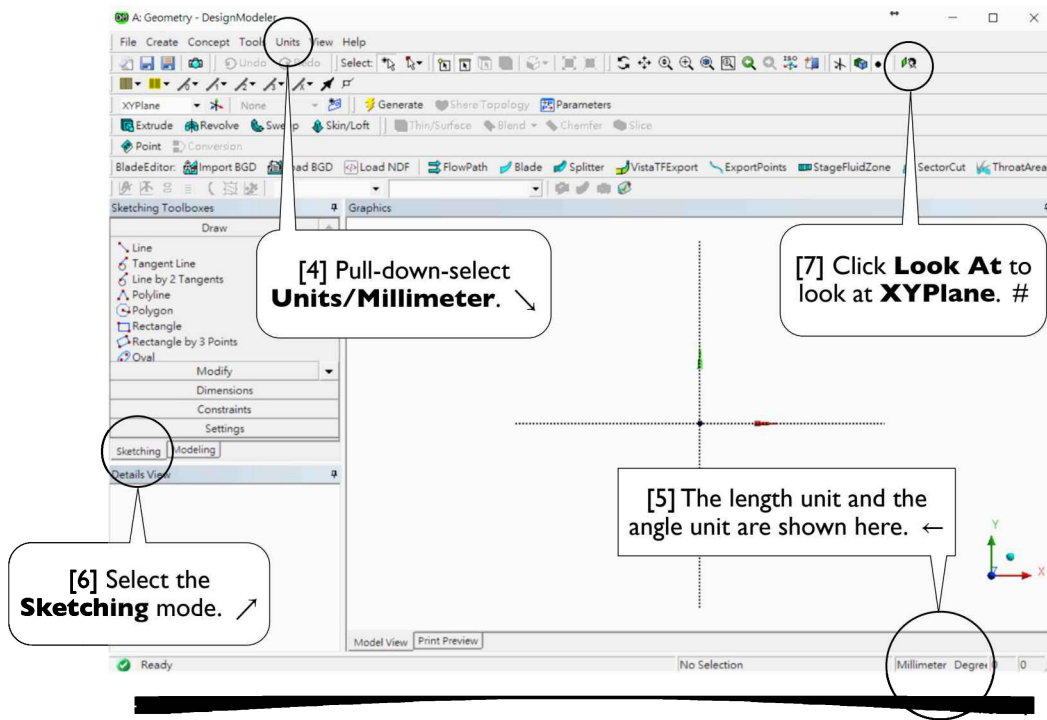


[1] Double-click to launch Workbench. →

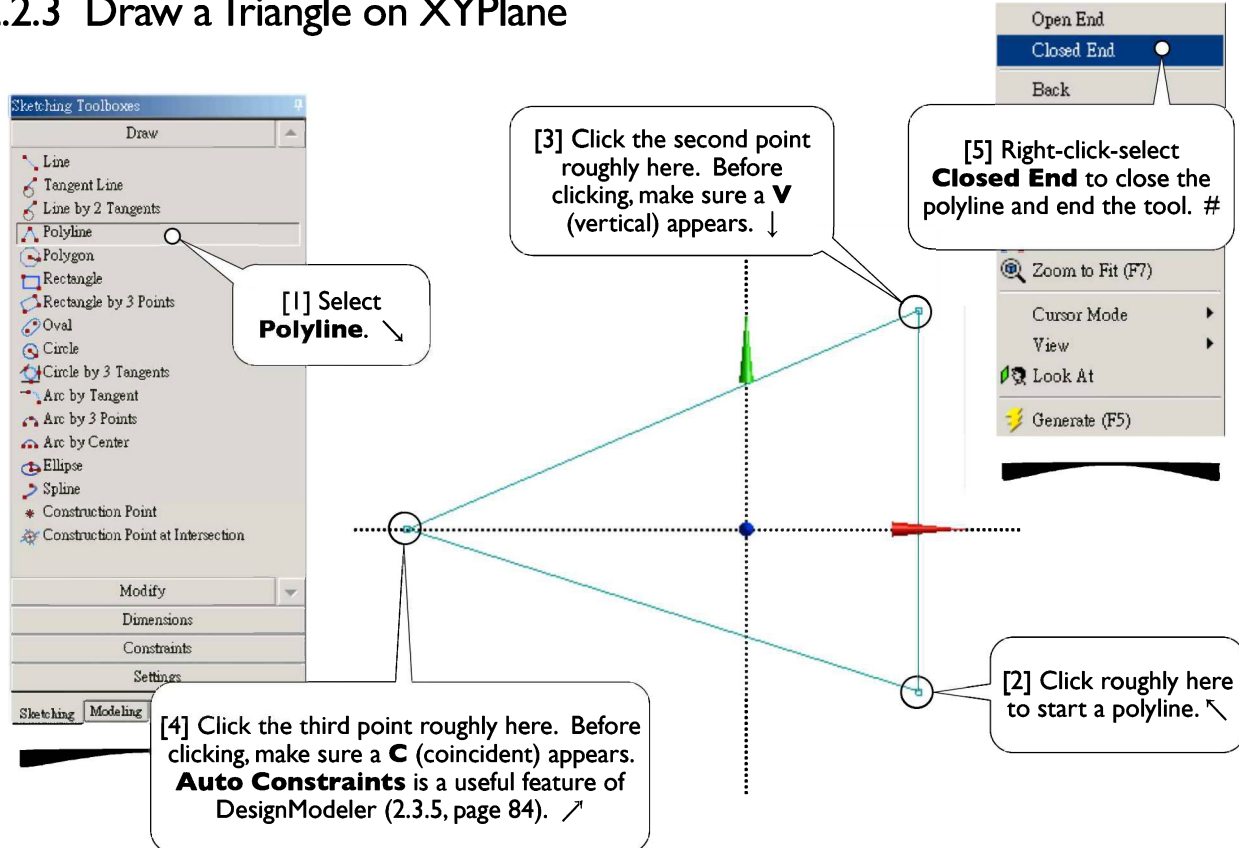


[3] Right-click the **Geometry** cell and select **New DesignModeler Geometry...** to start up DesignModeler. ↵

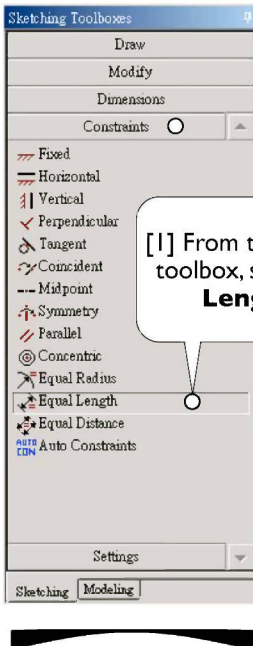
[2] Double-click to create a **Geometry** system (2.1.2[5], page 57). ↑



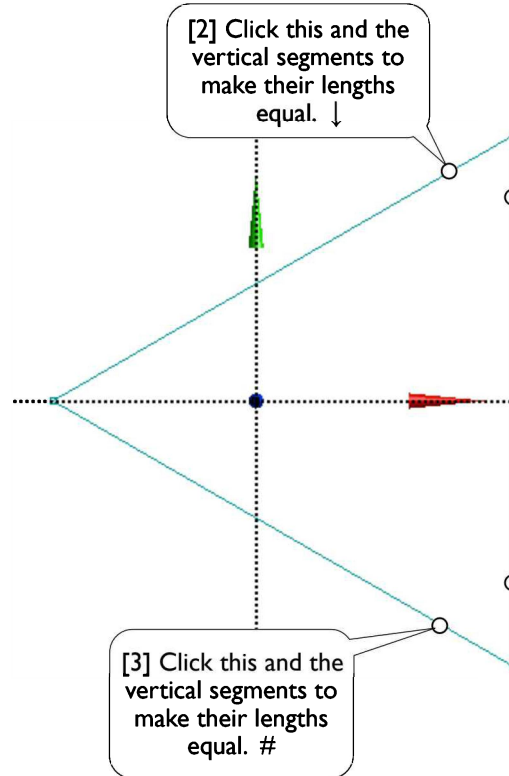
2.2.3 Draw a Triangle on XYPlane



2.2.4 Make the Triangle Regular

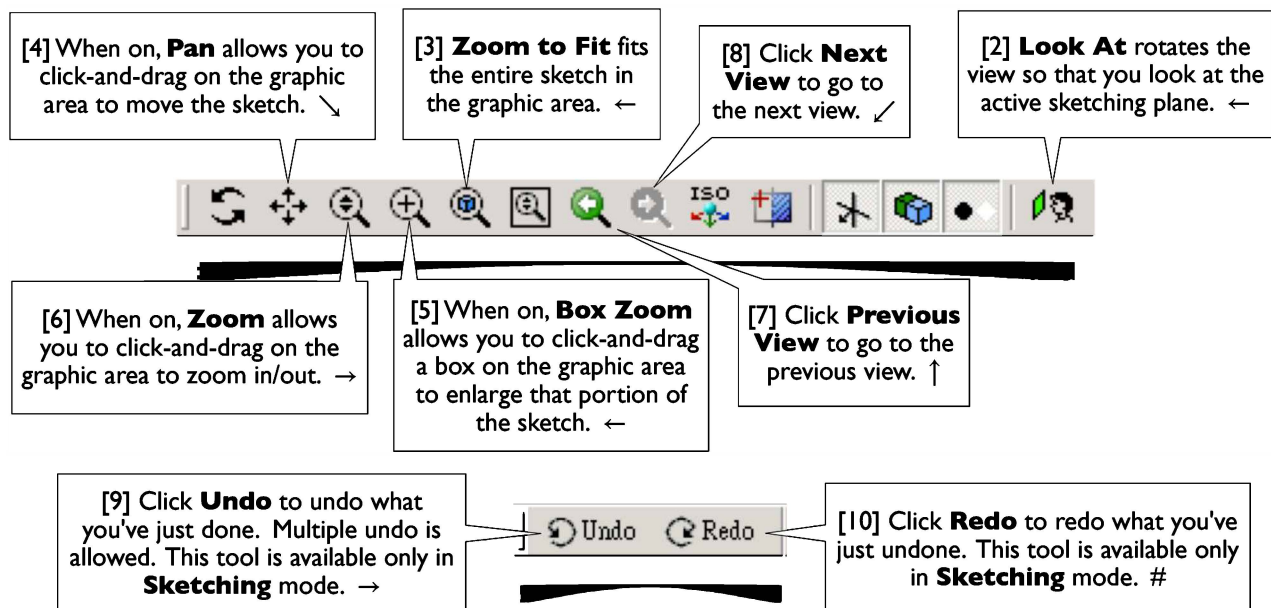


[1] From the **Constraints** toolbox, select the **Equal Length** tool. ↗

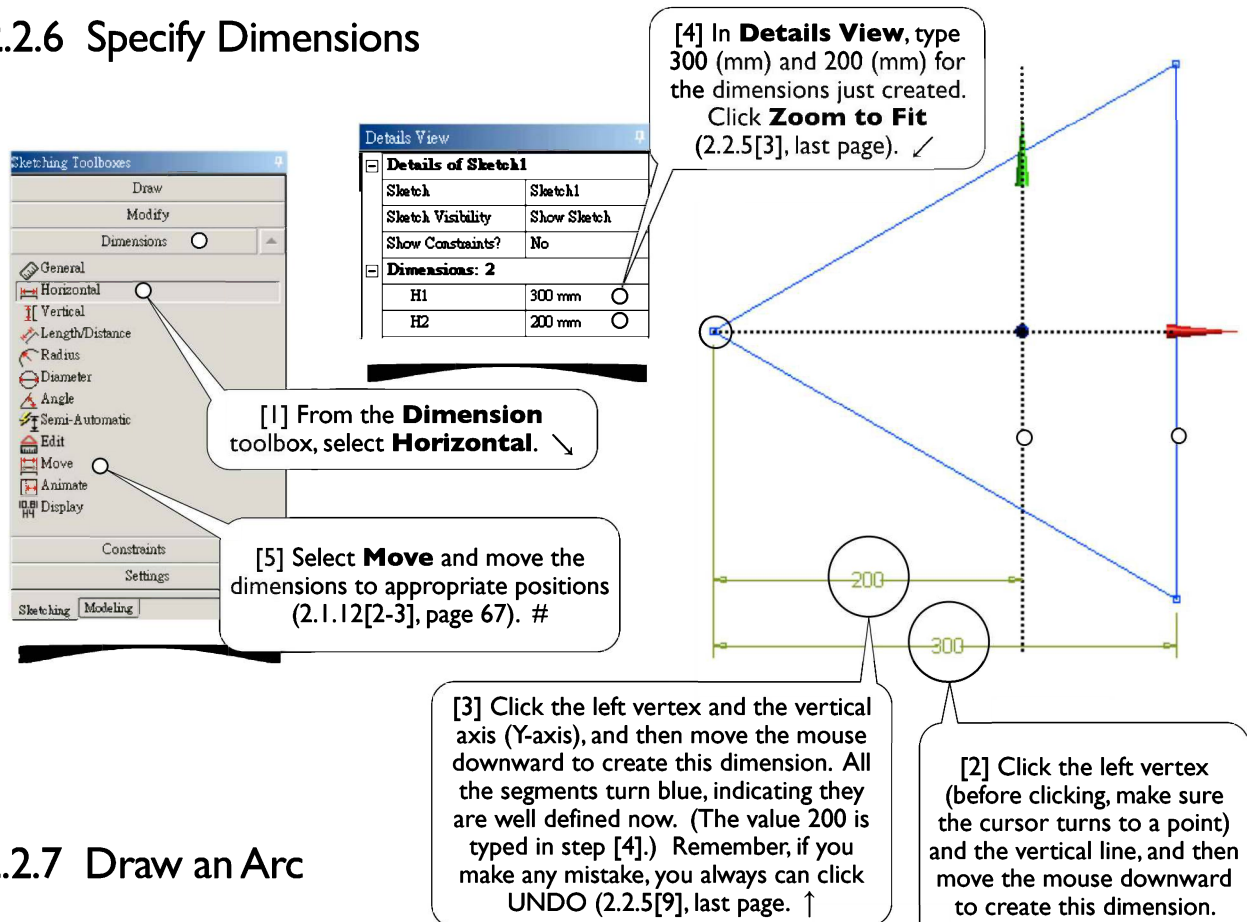


2.2.5 2D Graphics Controls

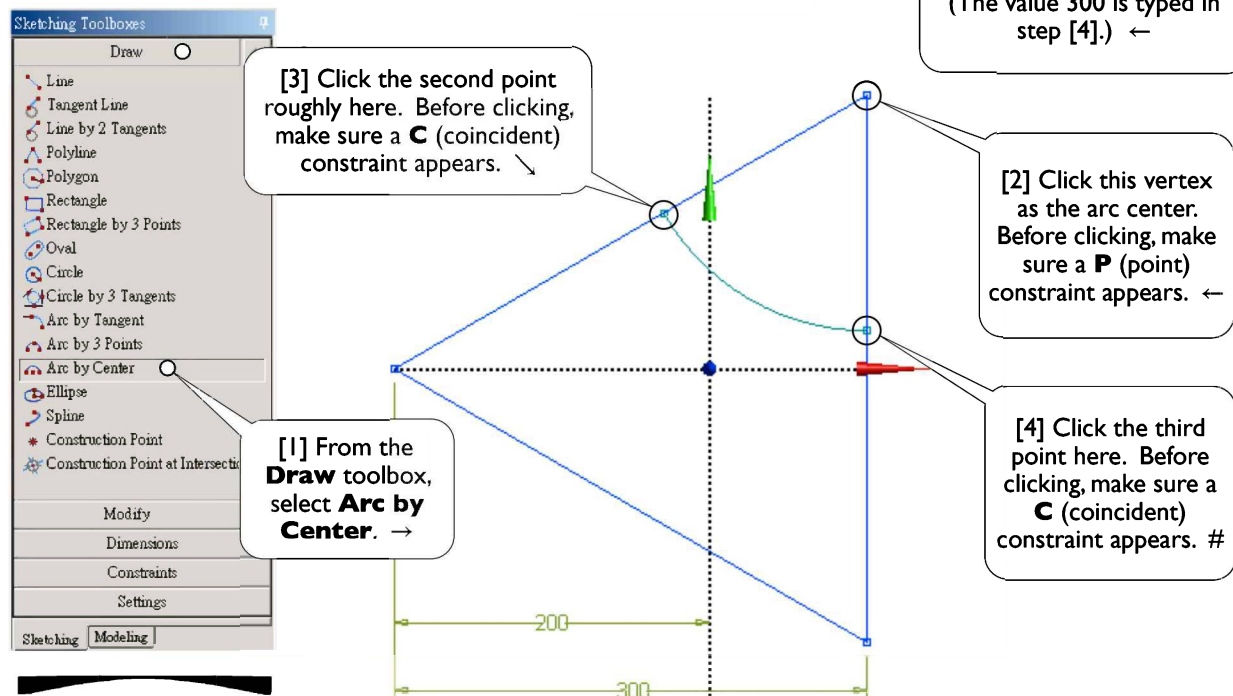
[1] Tools for 2D graphics controls are available in the **Display Toolbar** [2-10]. Click the tools in [4-6] to toggle them on/off. Feel free to use these tools any time. Try to click each tool now; they don't modify the model. Note that other ways to **Pan**, **Zoom**, and **Box Zoom** are given in 2.1.7[5-7] (page 64) and 2.3.4[1] (page 83). ↓



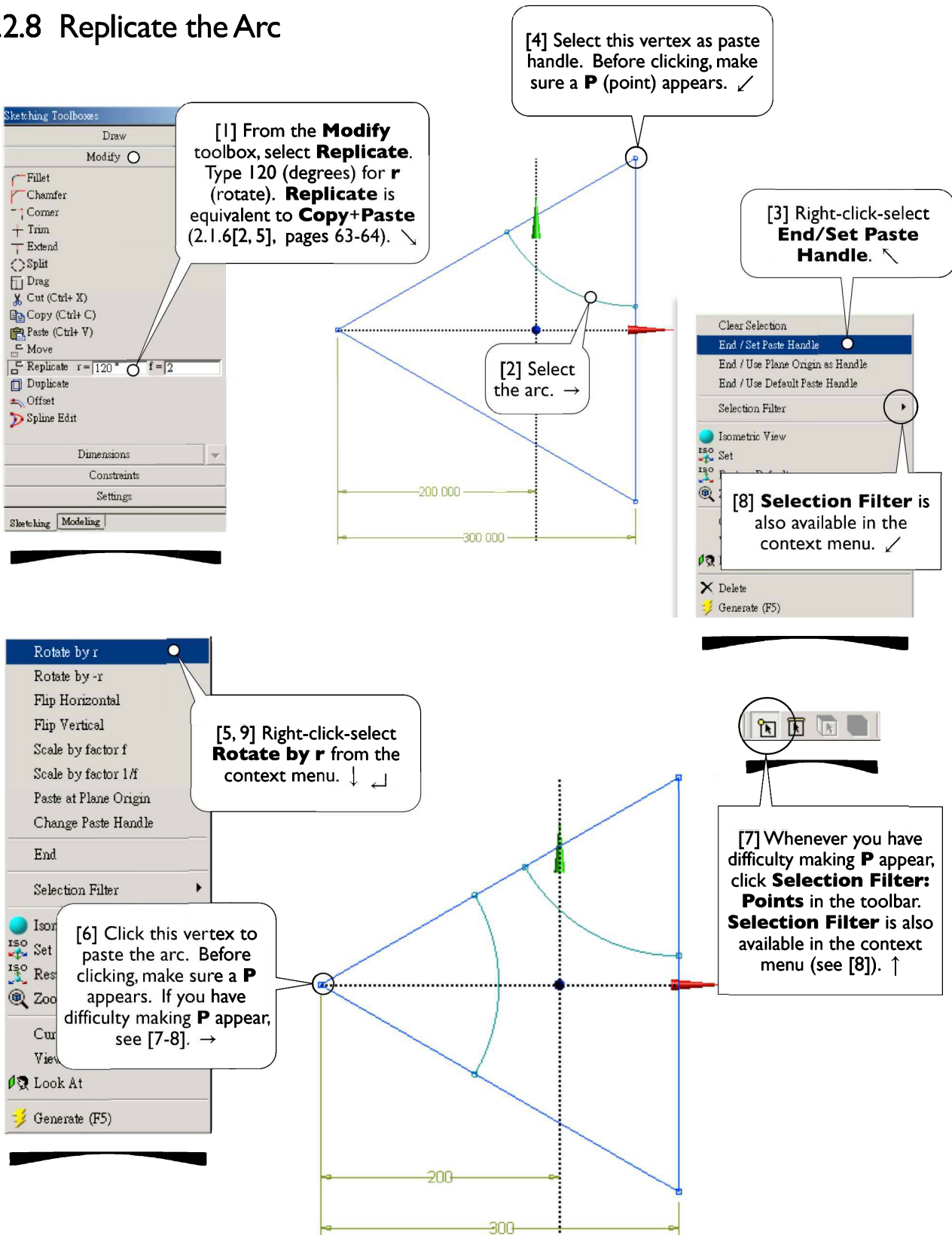
2.2.6 Specify Dimensions

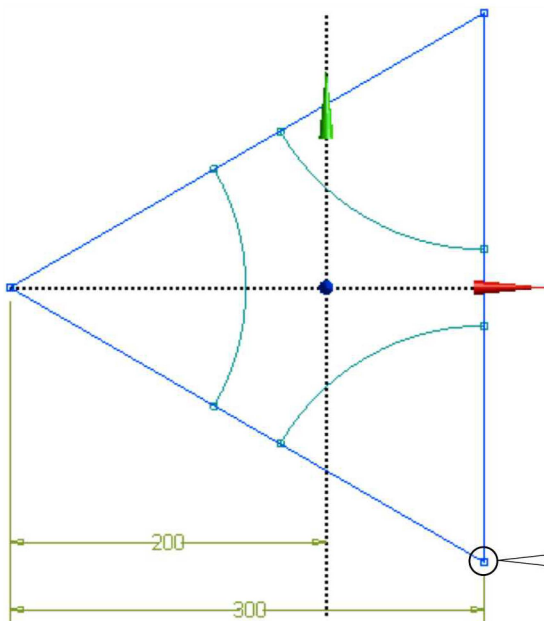


2.2.7 Draw an Arc

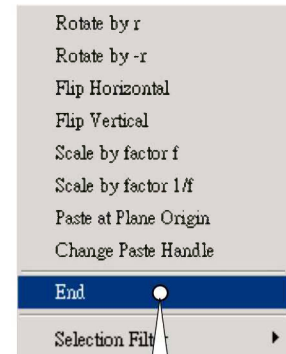


2.2.8 Replicate the Arc





[10] Select this vertex to paste the arc. Before clicking, make sure a **P** appears. →

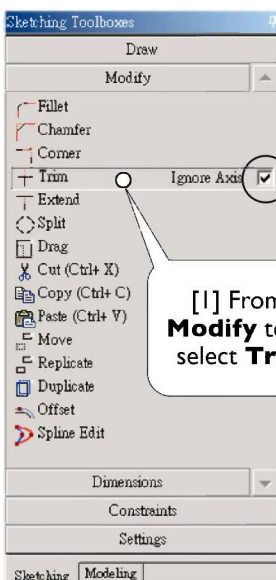


[11] Right-click-select **End** in the context menu to end the **Replicate** tool. Alternatively, press **ESC** to end the tool. ✓



[12] We chose to manually set the paste handle at a vertex ([3-4], last page) because we want to demonstrate the use of **Set Paste Handle** [3]. In this case, select **Use Plane Origin as Handle, Rotate by r**, and then **Paste at Plane Origin** have the same result. #

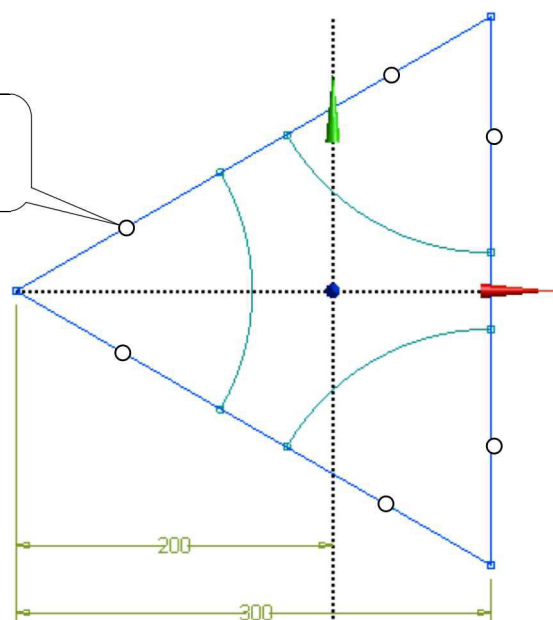
2.2.9 Trim Away Unwanted Segments



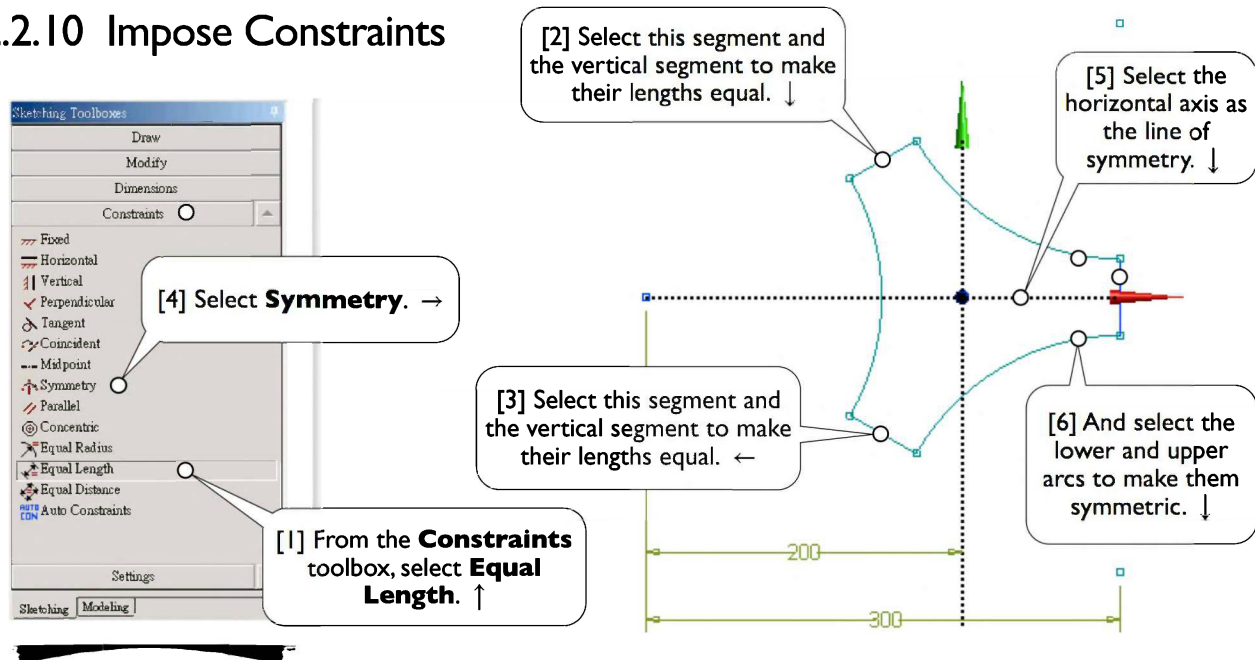
[2] Turn on **Ignore Axis** (2.1.8[2], page 65). ↓

[3] Click to trim away these 6 segments. #

[1] From the **Modify** toolbox, select **Trim**. ↑



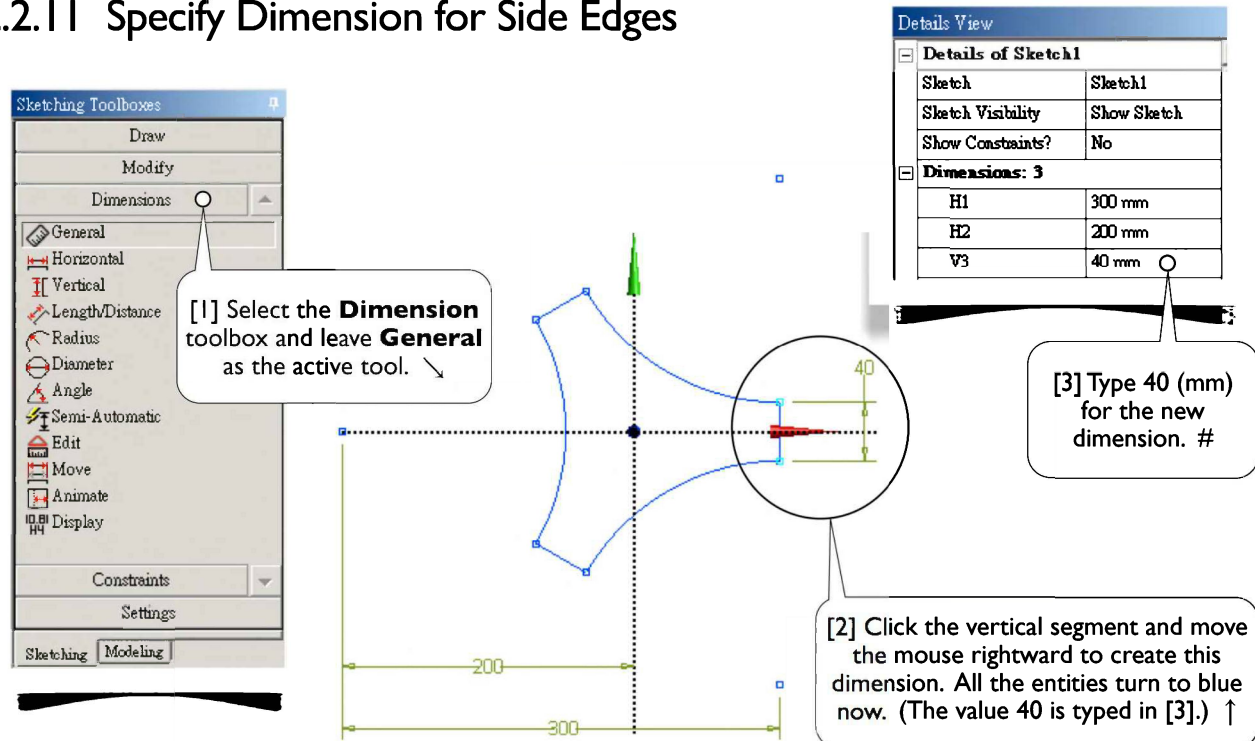
2.2.10 Impose Constraints



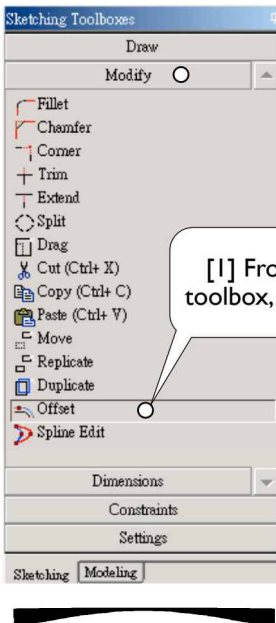
Constraint Status

[7] The three straight segments turn blue, indicating they are well-defined, while the three arcs remain greenish-blue, indicating they are not well-defined yet (under-constrained). Other color codes are black for fixed, red for over-constrained, and gray for inconsistency. #

2.2.11 Specify Dimension for Side Edges

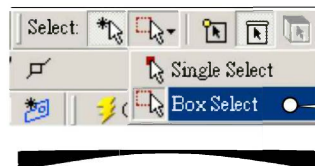
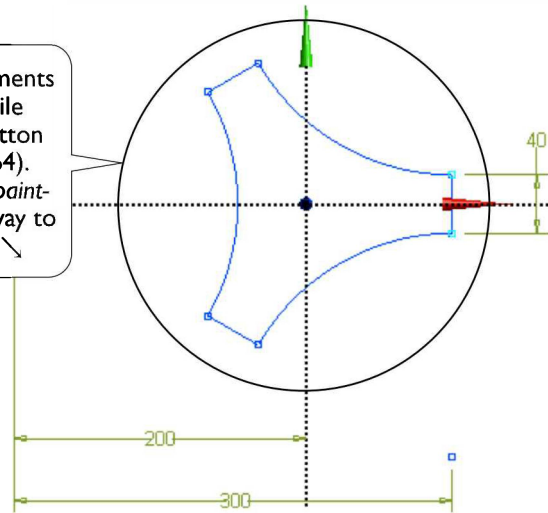


2.2.12 Create Offset



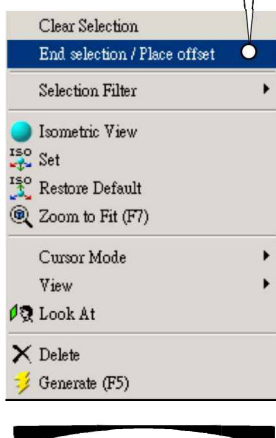
[1] From the **Modify** toolbox, select **Offset**. ↑

[2] Sweep-select all the segments (sweep each segment while holding your left mouse button down, see 2.1.7[3], page 64). Sweep-select is also called *paint-select*. (See [3] for another way to select multiple entities.) ↘

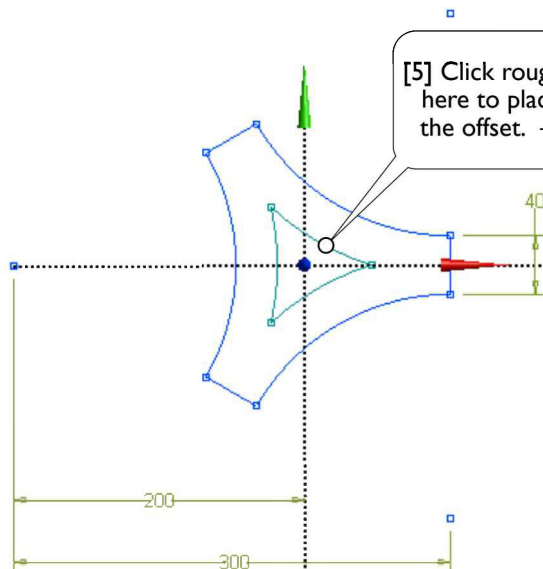


[3] Another way to select multiple entities is to switch **Select Mode** to **Box Select**, and then left-click-drag a box to select the entities inside the box. ←

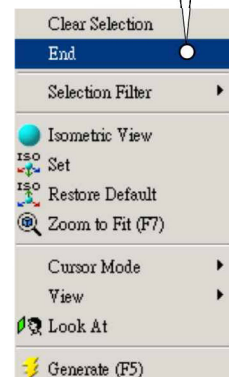
[4] Right-click-select **End selection/Place Offset** from the context menu. ↘

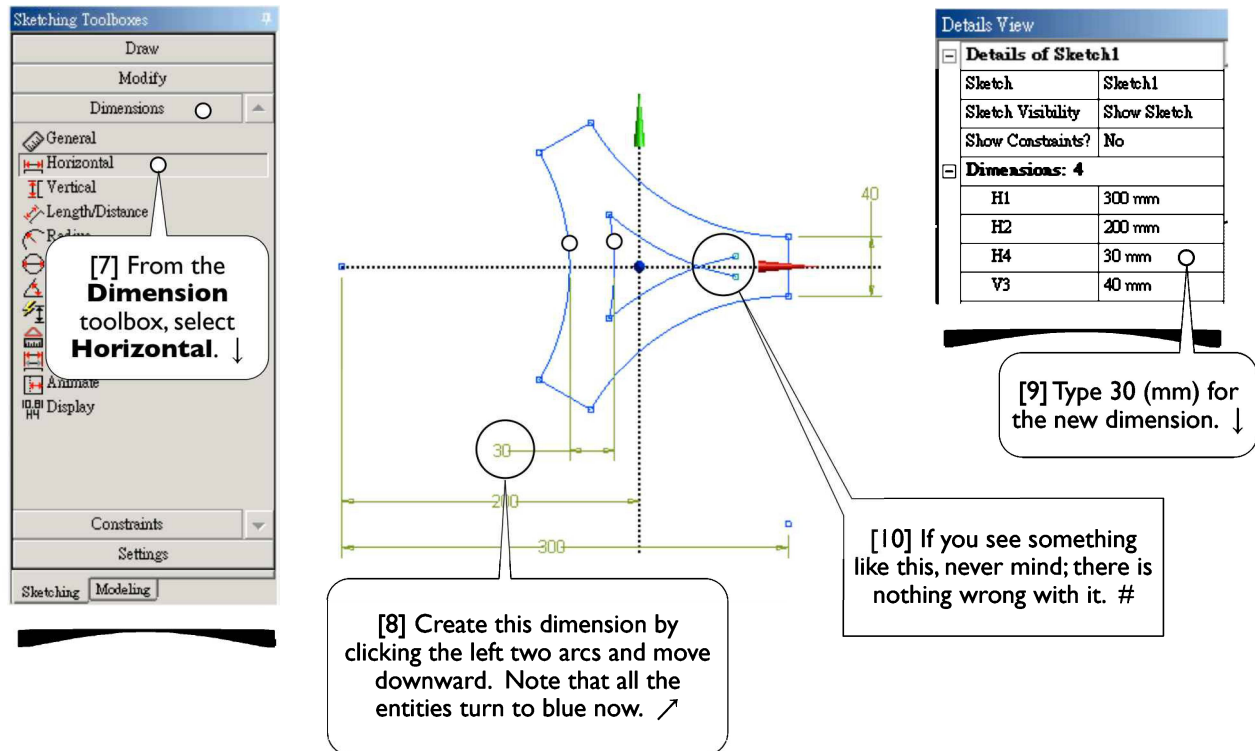


[5] Click roughly here to place the offset. →

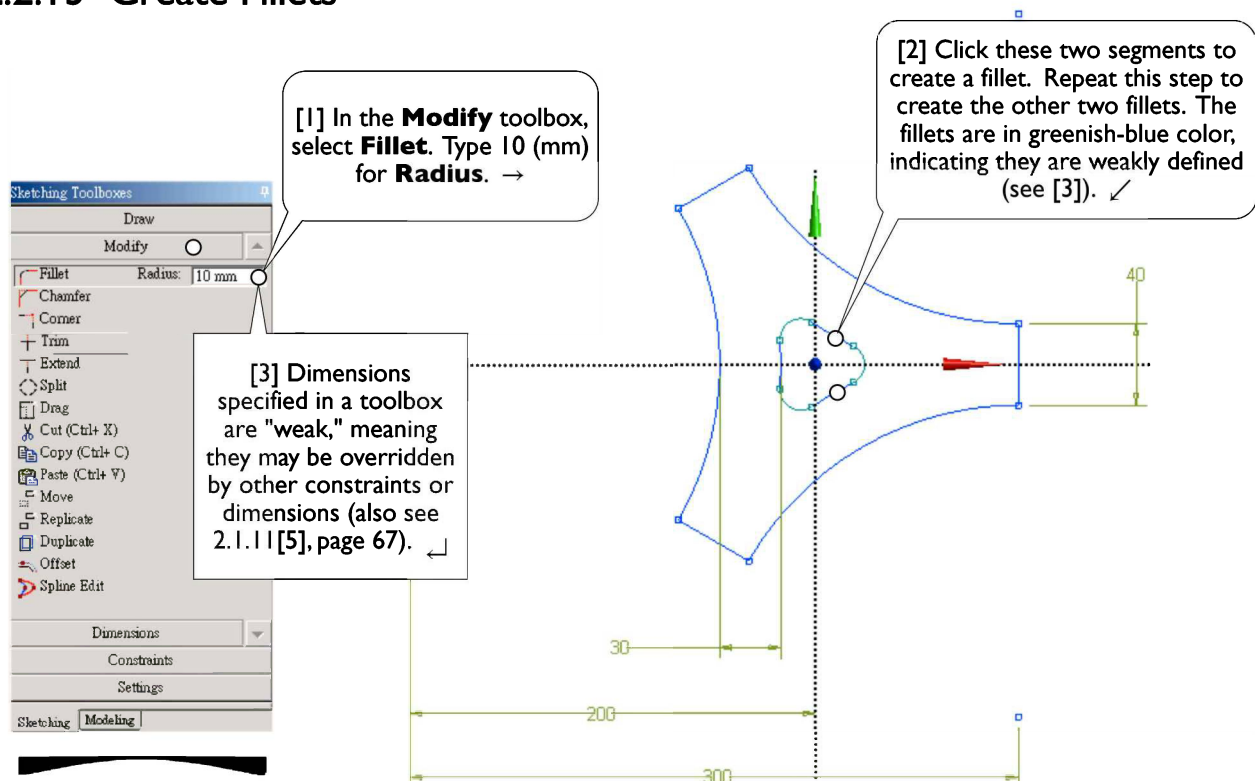


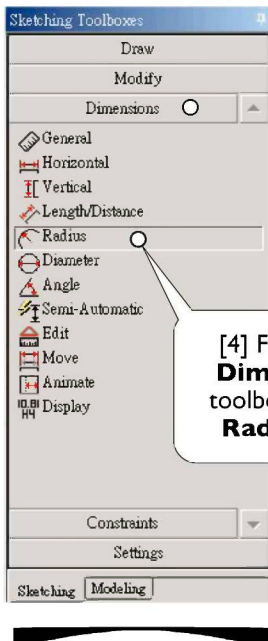
[6] Right-click-select **End** in the context menu, or press **ESC**, to close the **Offset** tool. ↵





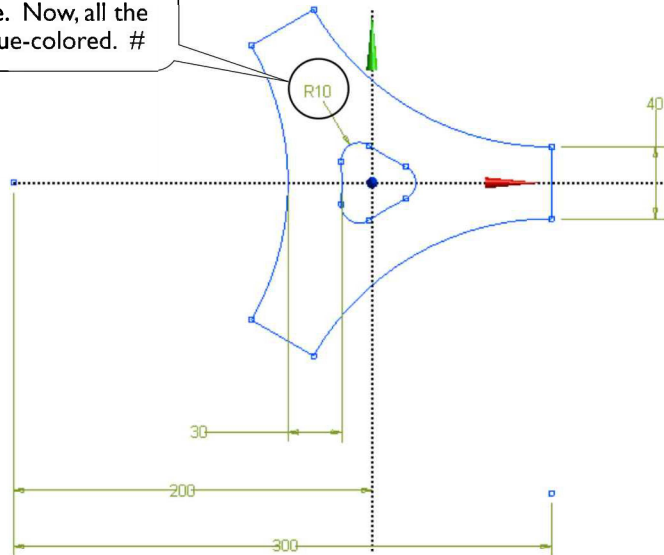
2.2.13 Create Fillets



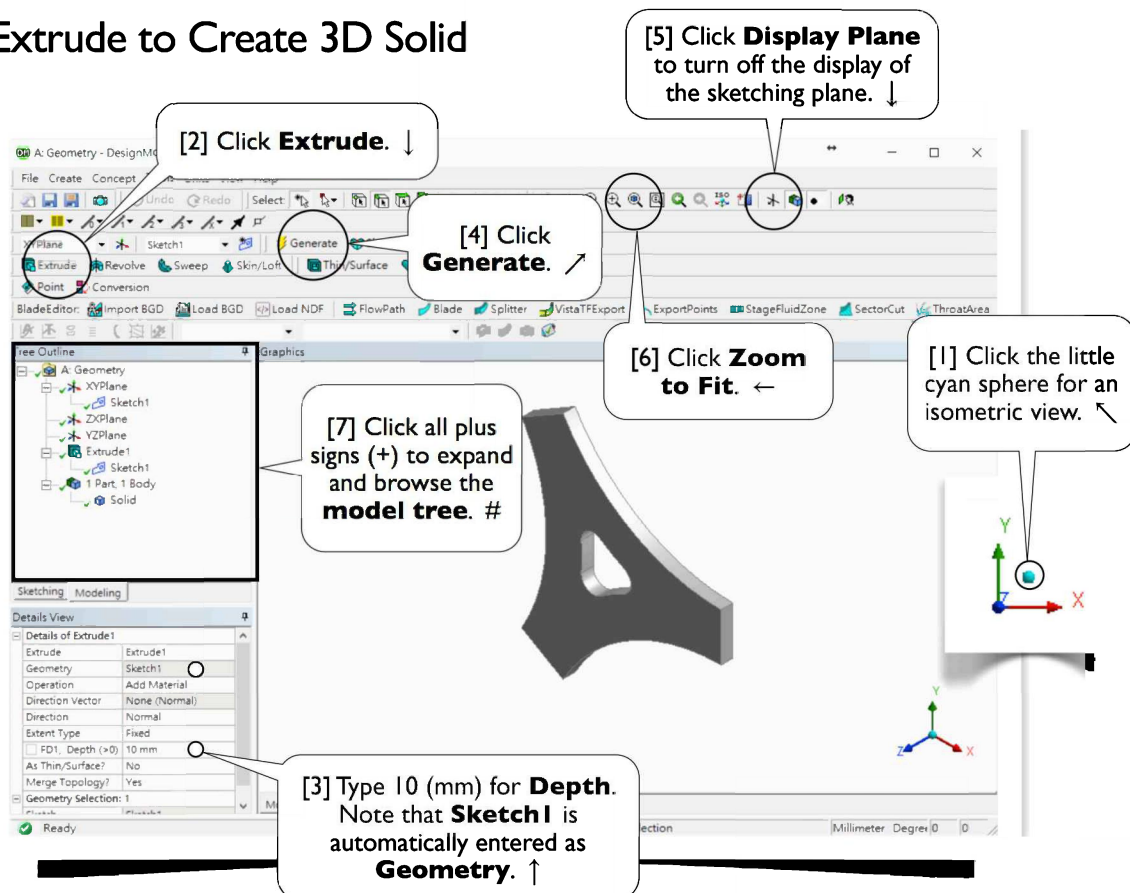


[4] From the **Dimension** toolbox, select **Radius**. ↗

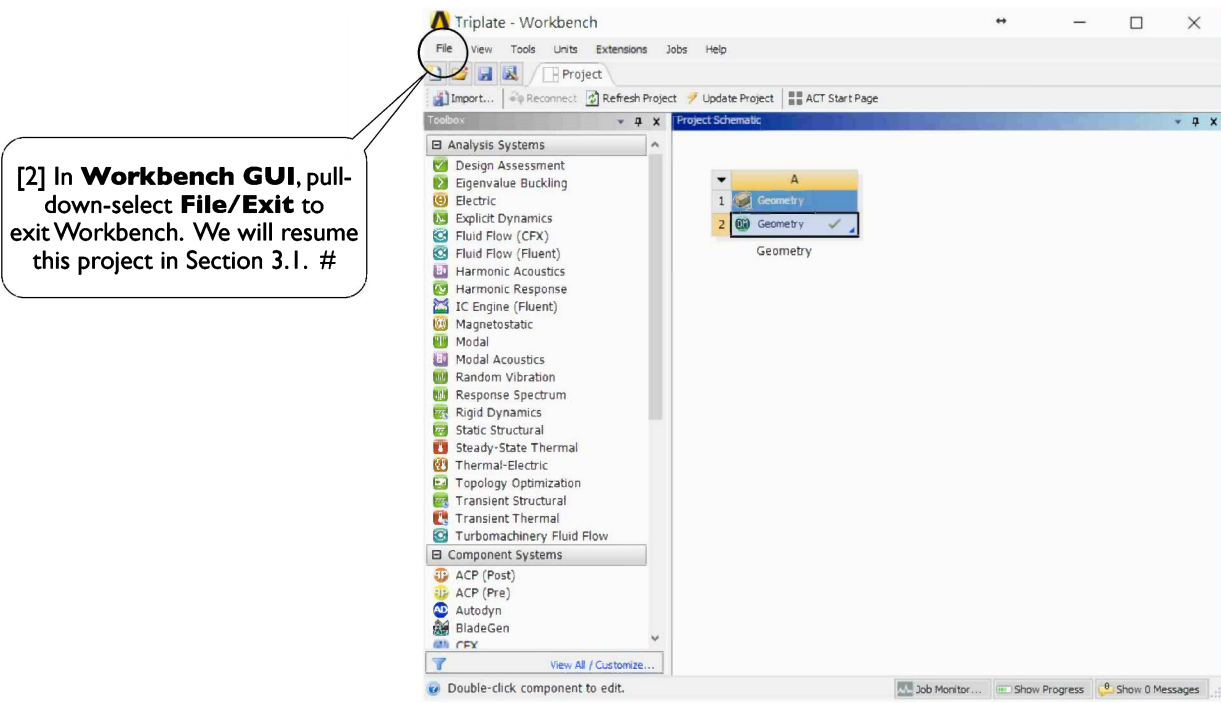
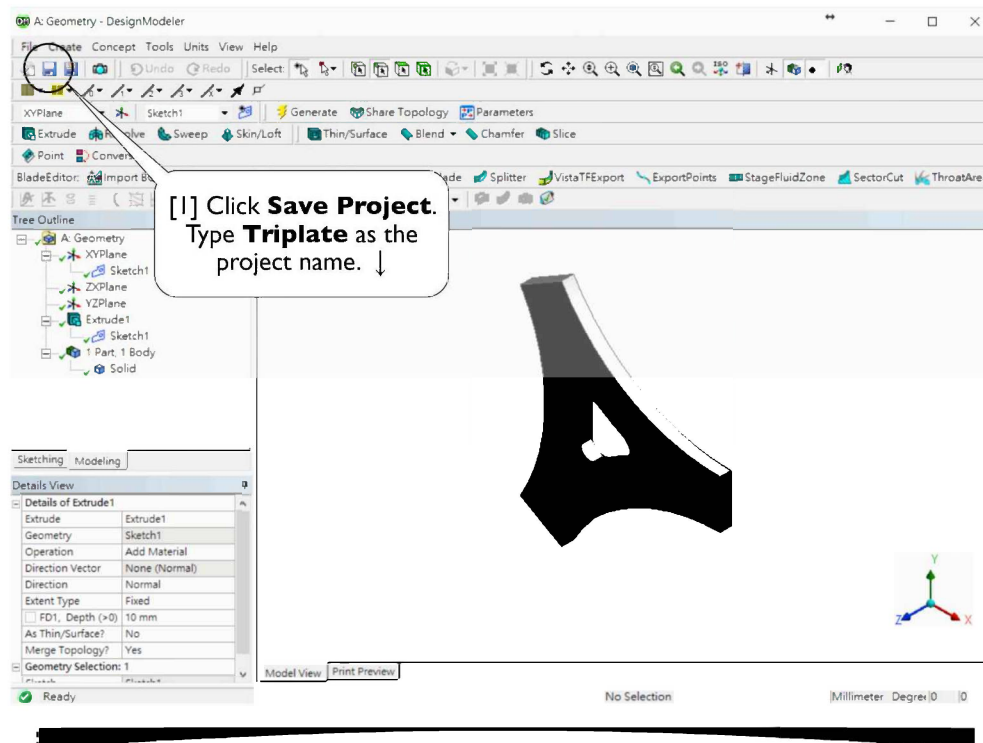
[5] Click one of the fillets to create this dimension. This turns a "weak" dimension to a "strong" one. Now, all the fillets are blue-colored. #



2.2.14 Extrude to Create 3D Solid

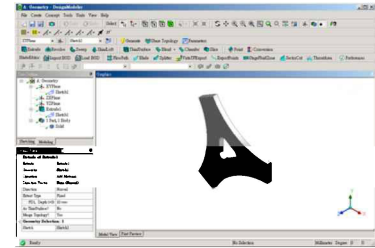


2.2.15 Save the Project and Exit Workbench



Section 2.3

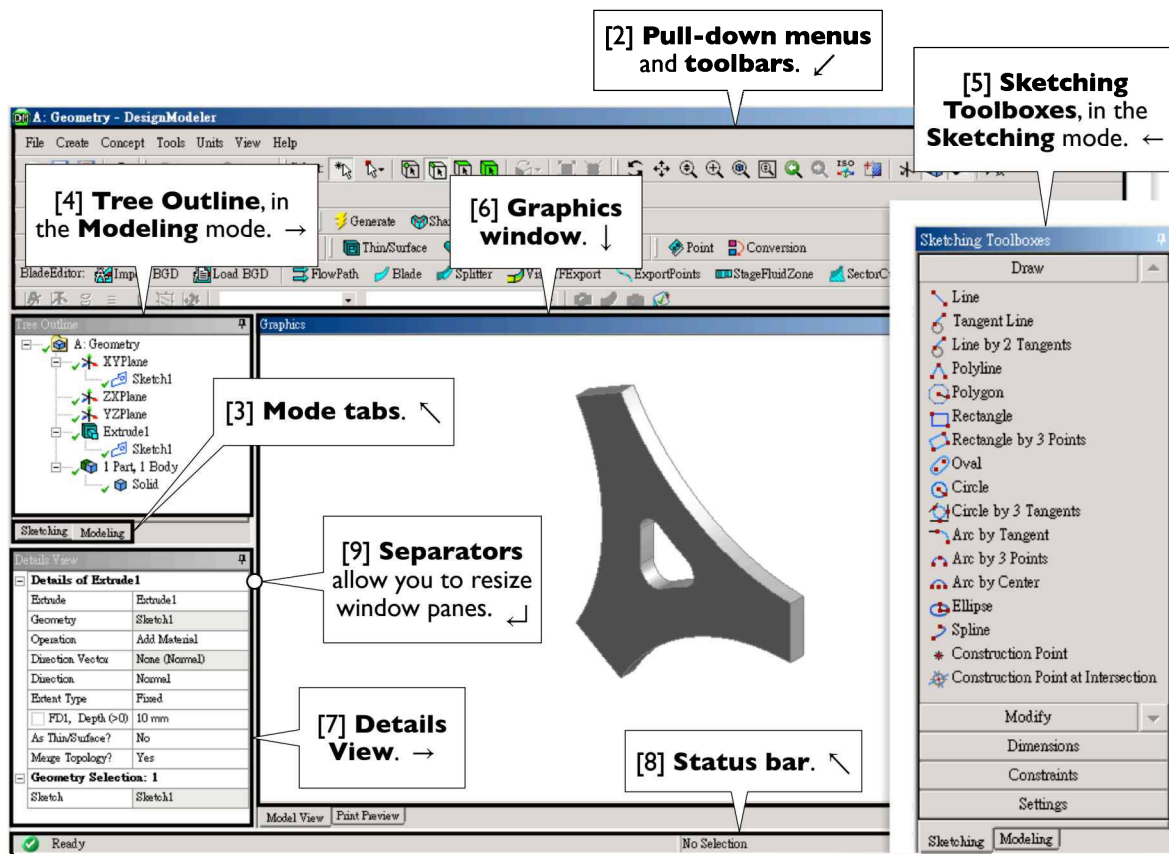
More Details



2.3.1 DesignModeler GUI

[1] **DesignModeler GUI** consists of several areas [2-8]. On the top are **pull-down menus** and **toolbars** [2]. On the bottom is a **status bar** [8]. In-between are several **window panes**. **Separators** [9] between window panes can be dragged to resize the window panes. You can make a window pane "float" by dragging or double-clicking its title bar. To return to its original position, simply double-click its title bar again.

Tree Outline [4] shares the same area with **Sketching Toolboxes** [5]. To switch between the **Modeling** mode and the **Sketching** mode, simply click a **mode tab** [3]. **Details View** [7] shows the detail information of the objects highlighted in **Tree Outline** [4] or **Graphics Window** [6]; the former displays a **Model Tree** (see [10], next page) while the latter displays a geometric model. Note that we discuss only the 2D functions of DesignModeler in this chapter and will discuss the 3D functions in Chapter 4. ↓



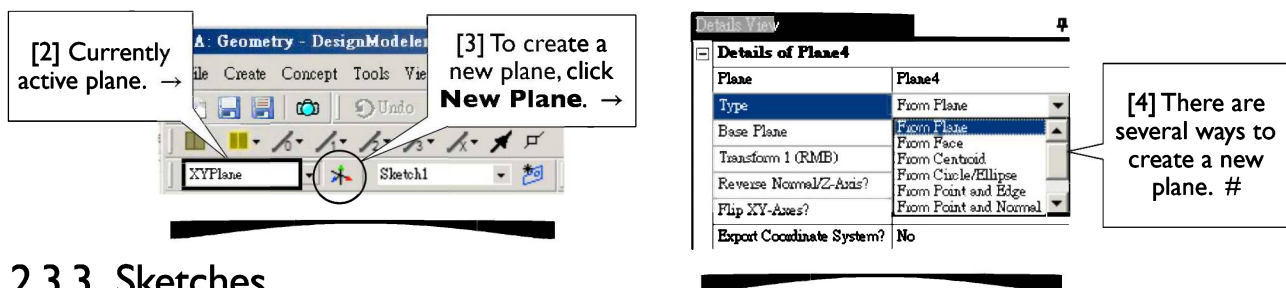
Model Tree

[10] **Tree Outline** [4] contains an outline of a **model tree**, the data structure of the geometric model. Each *branch* of the tree is called an **object**, which may contain one or more objects. At the bottom of the model tree is a **part** branch, which is the only object that will be exported to **Mechanical** for simulations. By right-clicking an object and selecting a tool from the *context menu*, you can operate on the object, such as delete, rename, duplicate, etc.

The order of the objects is relevant. DesignModeler renders the geometry according to the order of the objects in the model tree. New objects are normally added one after another. If you want to insert a new object **BEFORE** an existing object, right-click the existing object and select **Insert/...** from the context menu. After insertion, DesignModeler will re-render the geometry. #

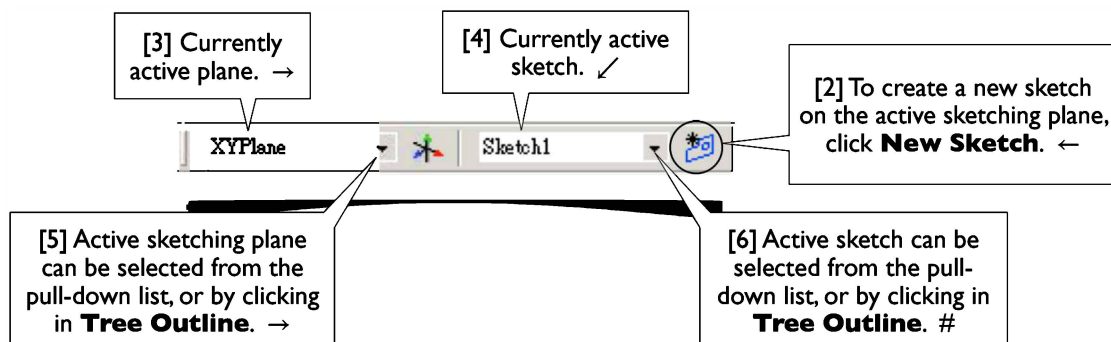
2.3.2 Sketching Planes

[1] A sketch must be created on a *sketching plane*, or simply called a *plane*; each plane may contain multiple sketches. In the beginning of a DesignModeler session, three planes preexist: **XYPlane**, **YZPlane**, and **ZXPlane**. The currently active plane is shown on the toolbar [2]. You can create as many new planes as needed [3]. There are several ways to create a new plane [4]. In this chapter, since we always sketch on **XYPlane**, we will not discuss the creation of sketching planes now and will discuss it in Chapter 4. ✓



2.3.3 Sketches

[1] A sketch consists of *points* and *edges*; an edge may be straight or curved. Dimensions and constraints may be imposed on points and edges. As mentioned (2.3.2[1]), multiple sketches may be created on a plane. To create a new sketch on an empty plane, you simply switch to **Sketching** mode and draw any geometric entities on it. Later, if you want to add a new sketch on that plane, you have to click **New Sketch** [2]. Exactly one plane and one sketch is active at a time [3-6]; newly created points and edges are added to the active sketch, and newly created sketches are added to the active plane. In this chapter, we almost exclusively work with a single sketch; the only exception is Section 2.6, in which a second sketch is created (2.6.4[3], page 106). When a new sketch is created, it becomes the active sketch. More on creating sketches will be discussed in Chapter 4. ✗



2.3.4 Sketching Toolboxes

[1] In the **Sketching** mode, five **Sketching Toolboxes** (2.3.1[5], page 81) are available: **Draw**, **Modify**, **Dimensions**, **Constraints**, and **Settings** [2-6]. Most of the tools in the toolboxes are self-explained. The best way to learn these tools is to try them out individually. During the tryout, whenever you want to clean up the graphics window, pull-down-select **File/Start Over**. These sketching tools will be briefly discussed, starting from 2.3.6. Before we discuss these sketching tools, let's reiterate some useful tips about sketching as follows.

Pan, Zoom, and Box Zoom

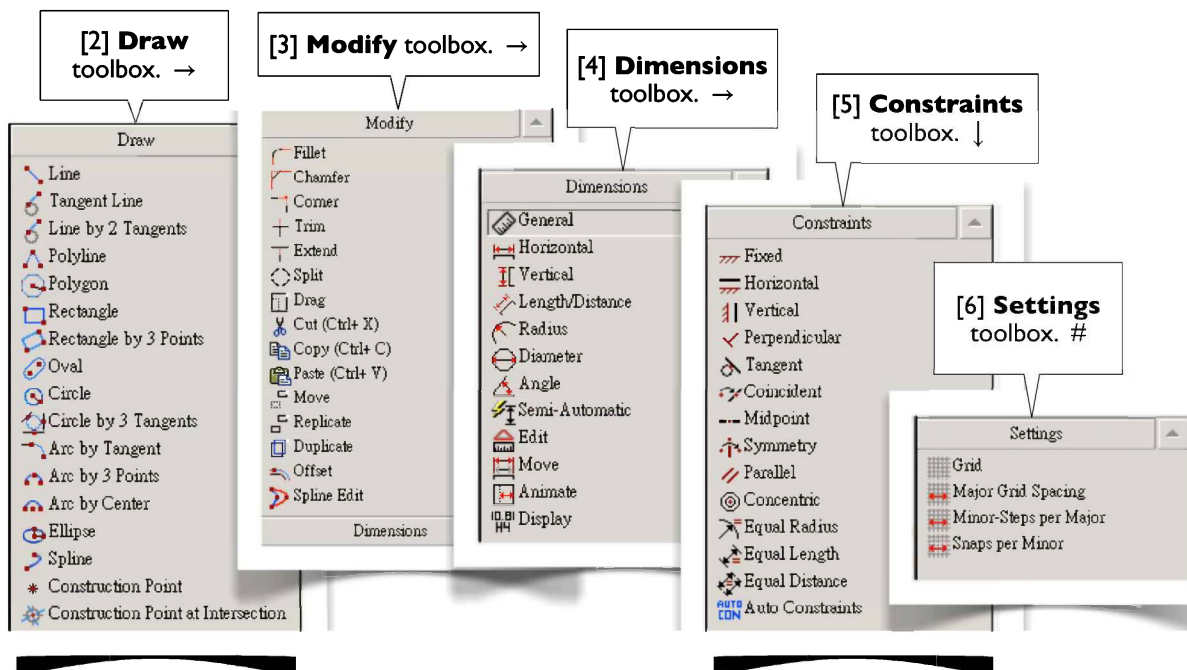
Besides the **Pan** tool in 2.2.5[4], page 72, a sketch can also be panned by dragging your mouse while holding down both the control key and the middle mouse button (2.1.7[7], page 64). Besides the **Zoom** tool in 2.2.5[6], page 72, a sketch can also be zoomed in/out by simply rolling your mouse wheel (2.1.7[6], page 64); the cursor position is the "zoom center." Besides the **Box Zoom** tool in 2.2.5[5], page 72, box zoom can also be done by dragging a rectangle in the graphics window using the right mouse button (2.1.7[5], page 64). After you are familiar with these mouse shortcuts, you usually don't need the **Pan**, **Zoom**, and **Box Zoom** tools in 2.2.5[4-6], page 72.

Context Menu

While most of the operations can be done by commands in pull-down menus or toolbars, many operations either require or are more efficient using a context menu. The context menu can be popped-up by right-clicking an entity in the graphics window or an object in the model tree. Try to explore whatever is available in the context menu.

Status Bar

The status bar (2.3.1[8], page 81) contains instructions on each operation. Look at the instructions whenever helpful. When a draw tool is in use, the coordinates of your mouse pointer are shown in the status bar. ✓

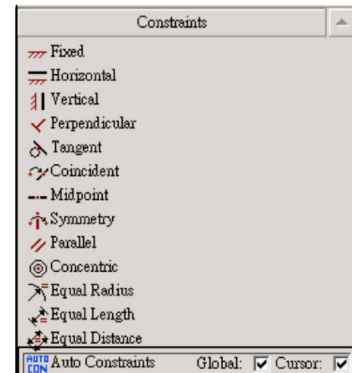


2.3.5 Auto Constraints^[Refs 1, 2]

[1] By default, DesignModeler is in the **Auto Constraints** mode, both globally and locally. DesignModeler attempts to detect the user's intentions and tries to automatically impose constraints on sketching entities. The following cursor symbols indicate the kind of constraints that are applied:

- C - The cursor is coincident with a line.
- P - The cursor is coincident with a point.
- T - The cursor is a tangent point.
- ⊥ - The cursor is a perpendicular foot.
- H - The line is horizontal.
- V - The line is vertical.
- // - The line is parallel to another line.
- R - The radius is equal to another radius.

Both **Global** and **Cursor** modes are based on all entities of the active plane (not just the active sketch). The difference is that **Cursor** mode only examines the entities nearby the cursor, while **Global** mode examines all the entities in the active plane. →



[2] By default, DesignModeler is in the **Auto Constraints** mode, both globally and locally. #

2.3.6 Draw Tools^[Ref 3]

Line

[2] Draws a line by two clicks.

Tangent Line

Click a point on a curve (e.g., circle, arc, ellipse, or spline) to create a line tangent to the curve at that point.

Line by 2 Tangents

Click two curves to create a line tangent to these two curves. Click a curve and a point to create a line tangent to the curve and connecting to the point.

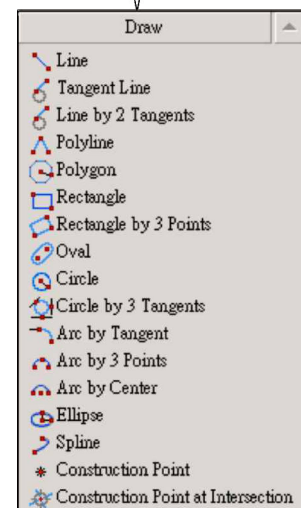
Polyline

A polyline consists of multiple straight lines. A polyline must be completed by choosing either **Open End** or **Closed End** from the context menu ([3], next page).

Polygon

Draws a regular polygon. The first click defines the center and the second click defines the radius of the circumscribing circle. ↵

[1] The **Draw** toolbox. ←



Rectangle by 3 Points

[4] The first two points define one side and the third point defines the other side.

Oval

The first two clicks define two centers, and the third click defines the radius.

Circle

The first click defines the center, and the second click defines the radius.

Circle by 3 Tangents

Select three edges (lines or curves) to create a circle tangent to these three edges.

Arc by Tangent

Click a point (usually an end point) on an edge to create an arc starting from that point and tangent to that edge; click a second point to define the other end and the radius of the arc.

Arc by 3 Points

The first two clicks define the two ends of the arc, and the third click defines a point between the ends.

Arc by Center

The first click defines the center, and two additional clicks define two ends.

Ellipse

The first click defines the center, the second click defines the major radius, and the third click defines the minor radius.

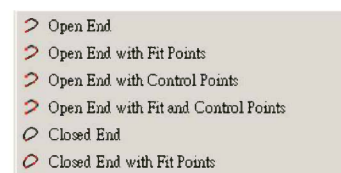
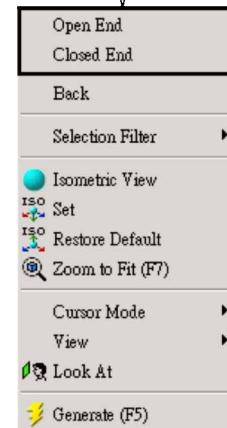
Spline

A spline is either rigid or flexible. A flexible spline can be edited or changed by imposing constraints, while a rigid spline cannot. After defining the last point, you must specify an ending condition [5]: either open end or closed end; either with fit points or without fit points.

Construction Point at Intersection

Select two edges; a construction point will be created at the intersection. ↗

[3] A polyline must be completed by choosing either **Open End** or **Closed End** from the context menu. ←



[5] A spline must be complete by specifying an ending condition from the context menu. ↓

How to delete edges?

[6] To delete edges, select them and choose **Delete** or **Cut** from the context menu. Multiple selection methods (e.g., control-selection or sweep-selection) can be used to select the edges. To clean up the graphics window entirely, pull-down-select **File/Start Over**. A general way of deleting any sketching entities (edges, dimensions, or constraints) is to right-click the entity in **Details View** and issue **Delete**. Also see 2.3.8[8-10], page 89, and 2.3.9[5-7], page 90.

How to abort a tool?

Simply press **ESC**. #

2.3.7 Modify Tools^[Ref 4]

Fillet

[2] Select two edges or a vertex to create a fillet. The radius of the fillet can be specified in the toolbox [3]. Note that this radius value is a weak dimension; i.e., it can be changed by other dimensions or constraints.

Chamfer

Select two edges or a vertex to create an equal-length chamfer. The sizes of the chamfer can be specified in the toolbox.

Corner

Select two edges, and the edges will be trimmed or extended up to the intersection point and form a sharp corner. The clicking points decide which sides to be trimmed.

Trim

Select an edge, and the portion of the edge will be removed up to its intersection with another edge, axis, or point.

Extend

Select an edge, and the edge will be extended up to an edge or axis.

Split

This tool splits an edge into several segments depending on the options from the context menu [4]. **Split Edge at Selection:** Click an edge, and the edge will be split at the clicking point. **Split Edges at Point:** Click a point, and all the edges passing through that point will be split at that point. **Split Edge at All Points:** Click an edge, and the edge will be split at all points on the edge. **Split Edge into n Equal Segments:** Click an edge and specify a value n , and the edge will be split equally into n segments.

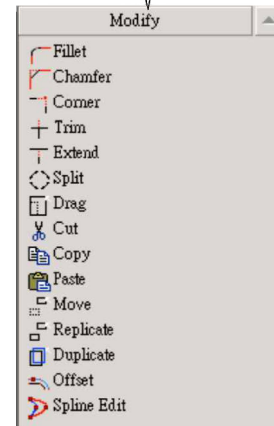
Drag

Drags a point or an edge to a new position. All the constraints and dimensions are preserved.

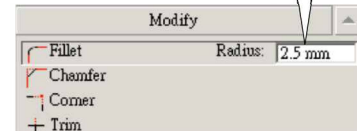
Copy

Copies the selected entities to a "clipboard." A **Paste Handle** must be specified using one of the methods in the context menu [5]. After completing this tool, **Paste** tool is automatically activated. ↗

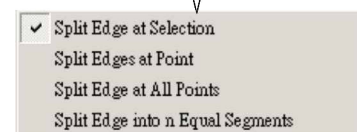
[1] The **Modify** toolbox. ←



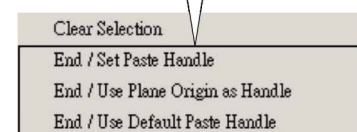
[3] The radius of fillets can be suggested; it is a "weak" dimension. ↓



[4] Options of **Split** in the context menu. ↓



[5] Options of **Copy** in the context menu. ↵



Cut

[6] Similar to **Copy**, except that the copied entities are removed.

Paste

Pastes the entities in the "clipboard" to the graphics window. The click defines the point at which the **Paste Handle** positions. Many options can be chosen from the context menu [7], where the rotating angle **r** and the scaling factor **f** can be specified in the toolbox.

Move

Equivalent to a **Cut** followed by a **Paste**. (The original is removed.)

Replicate

Equivalent to a **Copy** followed by a **Paste**. (The original is preserved.)

Duplicate

Similar to **Replicate**. However, **Duplicate** copies entities to the same position in the active plane. **Duplicate** can be used to copy features of a solid body or plane boundaries.

Offset

Creates a set of edges that are offset by a distance from an existing set of edges.

Spline Edit

Used to modify flexible splines. You can insert, delete, drag the fit points, etc [8]. For details, see the reference^[Ref 4]. ↗

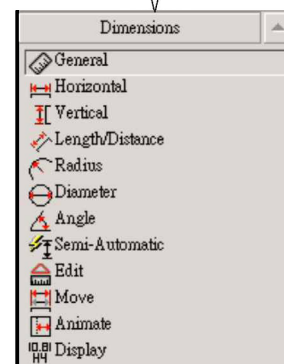
[7] Options of **Paste** in the context menu. ↓

Rotate by r Degrees
Rotate by -r Degrees
Flip Horizontal
Flip Vertical
Scale by factor f
Scale by factor 1/f
Paste at Plane Origin
Change Paste Handle

[8] Options of **Spline Edit** in the context menu. #

Select New Spline
Re-fit Spline
Create Missing Fit Points
Delete New Fit Points
Create Missing Control Points
Drag Fit Point
Drag Control Point
Insert Fit Point
Delete Fit Point

[1] The **Dimension** toolbox. ✓



2.3.8 Dimensions Tools^[Ref 5] [1]

General

[2] Allows creation of any of the dimension types, depending on what edge and context-menu options are selected. If the selected edge is a straight line, the default dimension is its length ([3], next page.) If the selected edge is a circle or arc, the default dimension is its radius ([4], next page). ↵

Horizontal

[5] Select two points to specify a horizontal. If you select an edge (instead of a point), the end point near the click will be picked.

Vertical

Similar to **Horizontal**.

Length/Distance

Select two points to specify a distance dimension. You also can select a point and a line to specify the distance between the point and the line.

Radius

Select a circle or arc to specify a radius dimension. If you select an ellipse, the major (or minor) radius will be specified.

Diameter

Select a circle or arc to specify a diameter dimension.

Angle

Select two lines to specify an angle. By varying the selection order and location, you can control which angle you are dimensioning. The end of the lines that you select will be the arrow point of the hands, and the angle is measured counterclockwise from the first hand to the second. If the angle is not what you want, repeatedly choose **Alternate Angle** from the context menu until a correct angle is selected [6].

Semi-Automatic

This tool displays a series of dimensions automatically to help you fully dimension the sketch.

Edit

Click a dimension and this tool allows you to change its name or values.

Move

Click a dimension and move it to a new position.

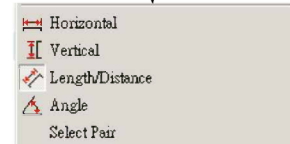
Animate

Click a dimension to show the animated effects.

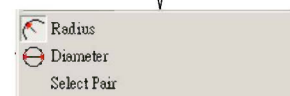
Display

Allows you to decide whether to display dimension names, values, or both. In this book, we always choose to display dimension values [7] rather than dimension names. ↗

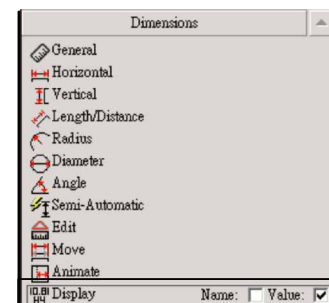
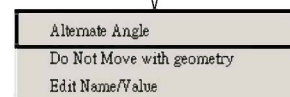
[3] Options of **General** in the context menu if you select a line. ↓



[4] Options of **General** in the context menu if you select a circle or arc. ←



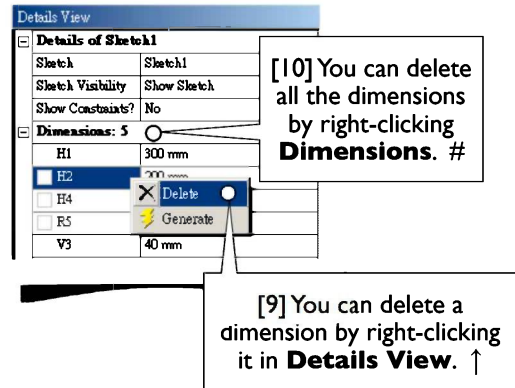
[6] You may repeatedly choose **Alternate Angle** from the context menu until the correct angle is selected. ↓



[7] In this book, we always display dimension values. ↙

How to delete dimensions?

[8] To delete a dimension, select the dimension in **Details View**, and choose **Delete** from the context menu [9]. You can delete ALL the dimensions by right-clicking **Dimensions** in **Details View** [10]. →



2.3.9 Constraints Tools^[Ref 6]

Fixed

[2] Applies on an edge to make it fully constrained if **Fix Endpoints** is selected [3]. If **Fix Endpoints** is not selected, then the edge's endpoints can be changed, but not the edge's position and slope.

Horizontal

Applies to a line to make it horizontal.

Vertical

Applies to a line to make it vertical.

Perpendicular

Applies to two edges to make them perpendicular to each other.

Tangent

Applies to two edges, one of which must be a curve, to make them tangent to each other.

Coincident

Select two points to make them coincident. Or, select a point and an edge to make the edge or its extension pass through the point. There are other possibilities, depending on how you select the entities.

Midpoint

Select a line and a point to make the midpoint of the line coincide with the point.

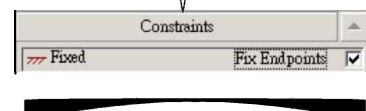
Symmetry

Select a line or an axis, as the line of symmetry, and then select two entities to make them symmetric about the line of symmetry. →

[1] The **Constraints** toolbox. ←



[3] If **Fix Endpoints** is selected, the edge will be fully constrained. ←



Parallel

[4] Applies to two lines to make them parallel to each other.

Concentric

Applies to two curves, which may be circle, arc, or ellipse, to make their centers coincident.

Equal Radius

Applies to two curves, which must be circle or arc, to make their radii equal.

Equal Length

Applies to two lines to make their lengths equal.

Equal Distance

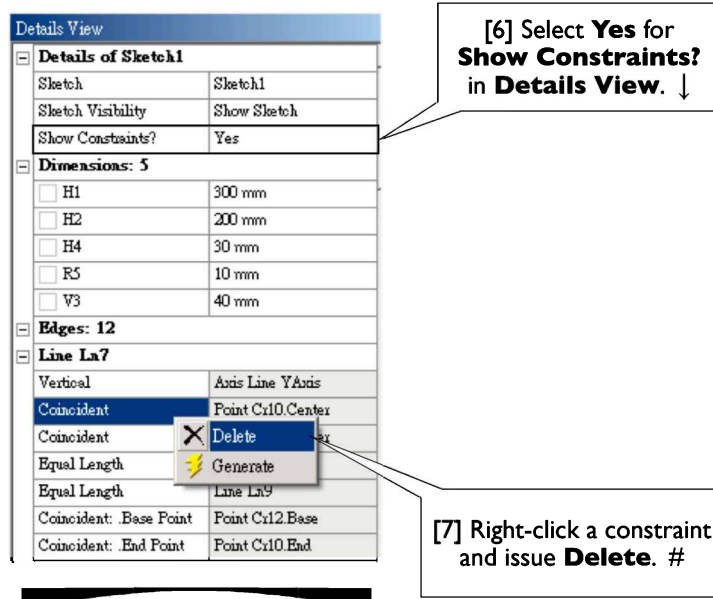
Applies to two distances to make them equal. A distance can be defined by selecting two points, two parallel lines, or one point and one line.

Auto Constraints

Allows you to turn on/off **Auto Constraints** (2.3.5, page 84). ↓

How to delete constraints?

[5] By default, constraints are not displayed in **Details View**. To display constraints, select **Yes** for **Show Constraints?** in **Details View** [6]. To delete a constraint, right-click the constraint and issue **Delete** [7]. ↓



2.3.10 Settings Tools^[Ref 7]

Grid

[2] Allows you to turn on/off grid visibility and snap capability [3-4]. The grid is not required to enable snapping.

Major Grid Spacing

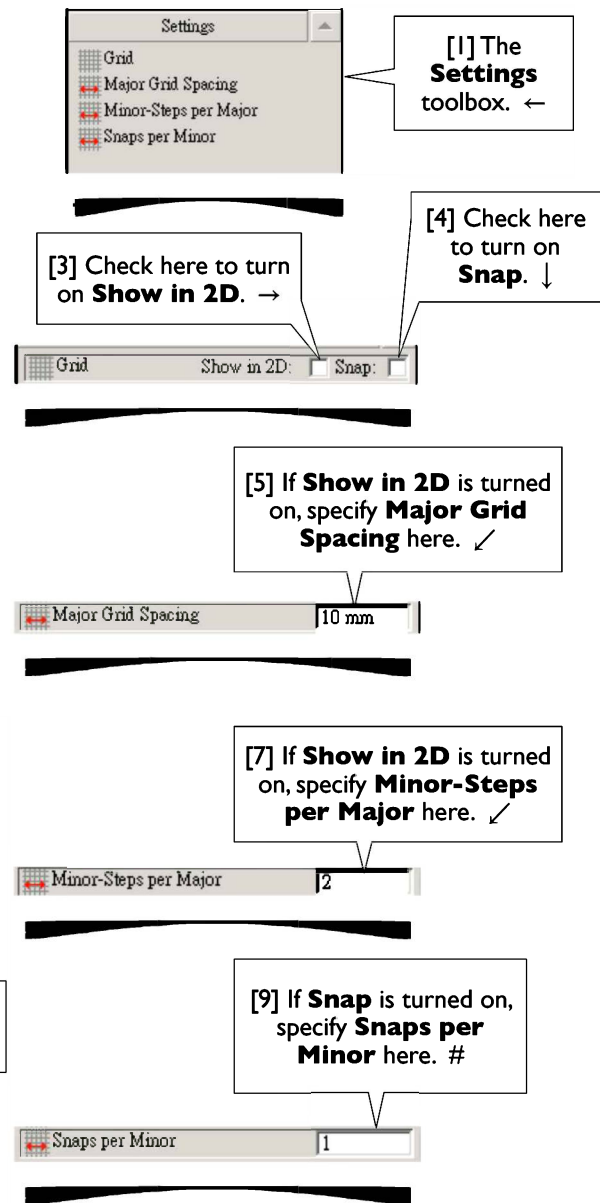
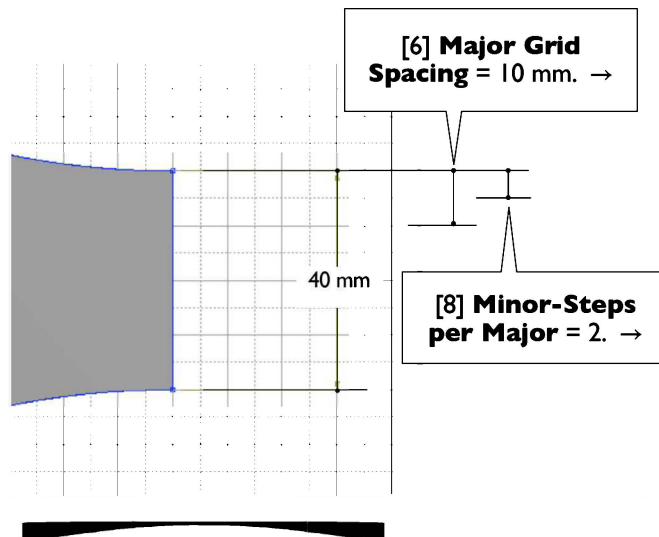
Allows you to specify **Major Grid Spacing** [5-6] if **Show in 2D** is turned on.

Minor-Steps per Major

Allows you to specify **Minor-Steps per Major** [7-8] if **Show in 2D** is turned on.

Snaps per Minor

Allows you to specify **Snaps per Minor** [9] if **Snap** is turned on. ↗

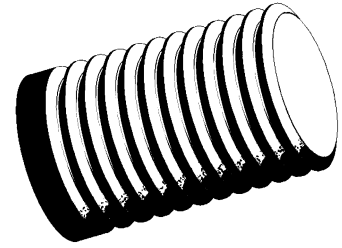


References

1. All Help>DesignModeler>ANSYS DesignModeler User's Guide>2D Sketching>Auto Constraints
2. All Help>DesignModeler>ANSYS DesignModeler User's Guide>2D Sketching>Constraints Toolbox//Auto Constraints
3. All Help>DesignModeler>ANSYS DesignModeler User's Guide>2D Sketching>Draw Toolbox
4. All Help>DesignModeler>ANSYS DesignModeler User's Guide>2D Sketching>Modify Toolbox
5. All Help>DesignModeler>ANSYS DesignModeler User's Guide>2D Sketching>Dimensions Toolbox
6. All Help>DesignModeler>ANSYS DesignModeler User's Guide>2D Sketching>Constraints Toolbox
7. All Help>DesignModeler>ANSYS DesignModeler User's Guide>2D Sketching>Settings Toolbox

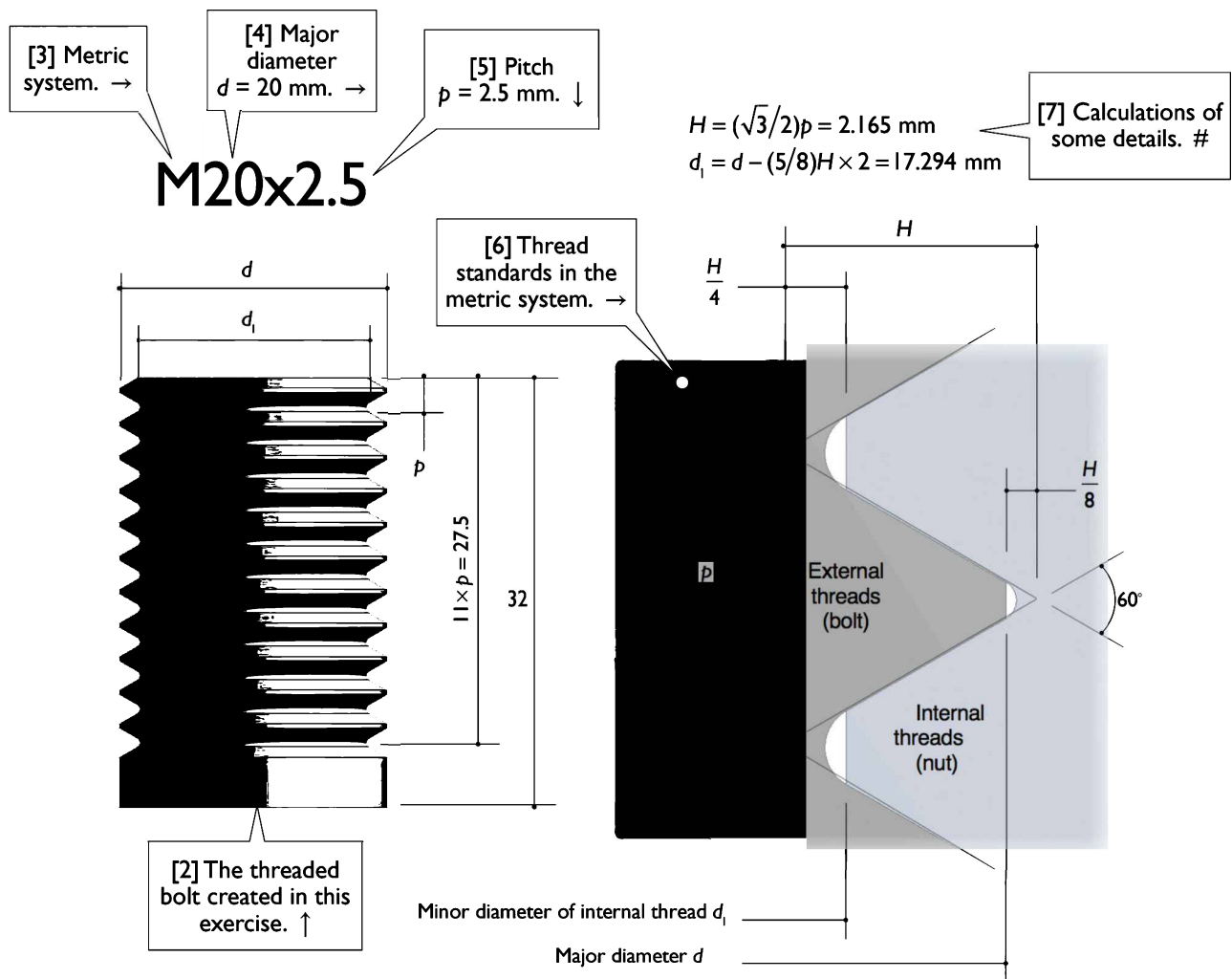
Section 2.4

M20x2.5 Threaded Bolt



2.4.1 About the M20x2.5 Threaded Bolt^[Refs 1, 2]

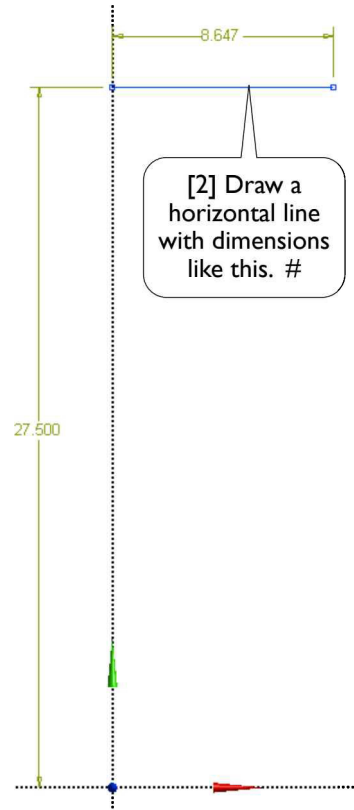
[1] In this section, we'll create a sketch and revolve the sketch 360° to generate a 3D solid body, a body representing a portion of an M20x2.5 threaded bolt as shown in [2-7]. We will use this sketch in Section 3.2 again to generate a 2D solid body, which is then used for a static structural simulation. ↓



2.4.2 Draw a Horizontal Line

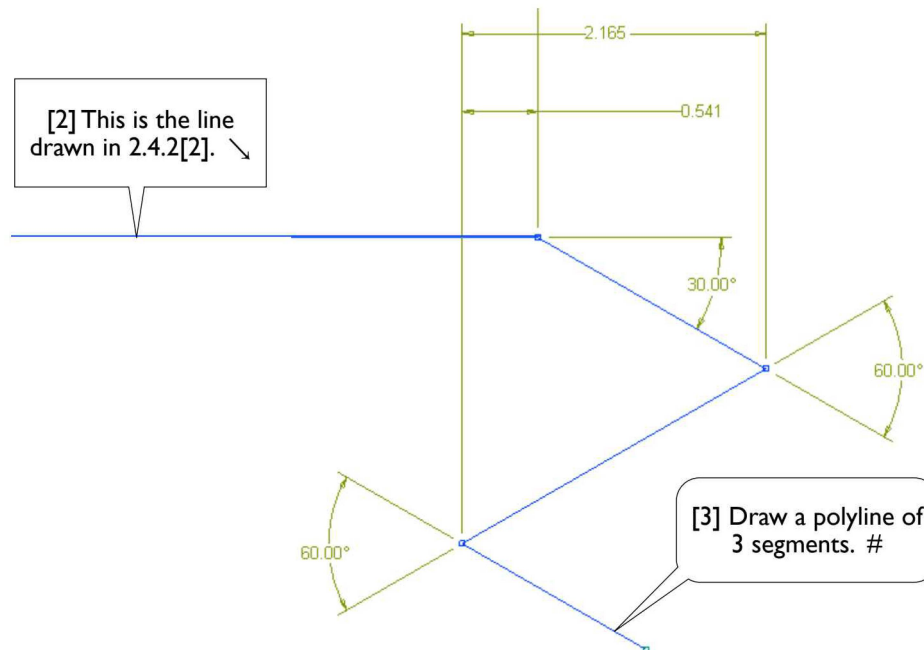
[1] Launch Workbench and create a **Geometry** system. Save the project as **Threads**. Start up DesignModeler. Select **Millimeter** as the length unit and make sure that **Degree** is the angle unit.

On **XYPlane**, draw a horizontal line and specify the dimensions (8.647 mm and 27.5 mm) as shown in [2]. →



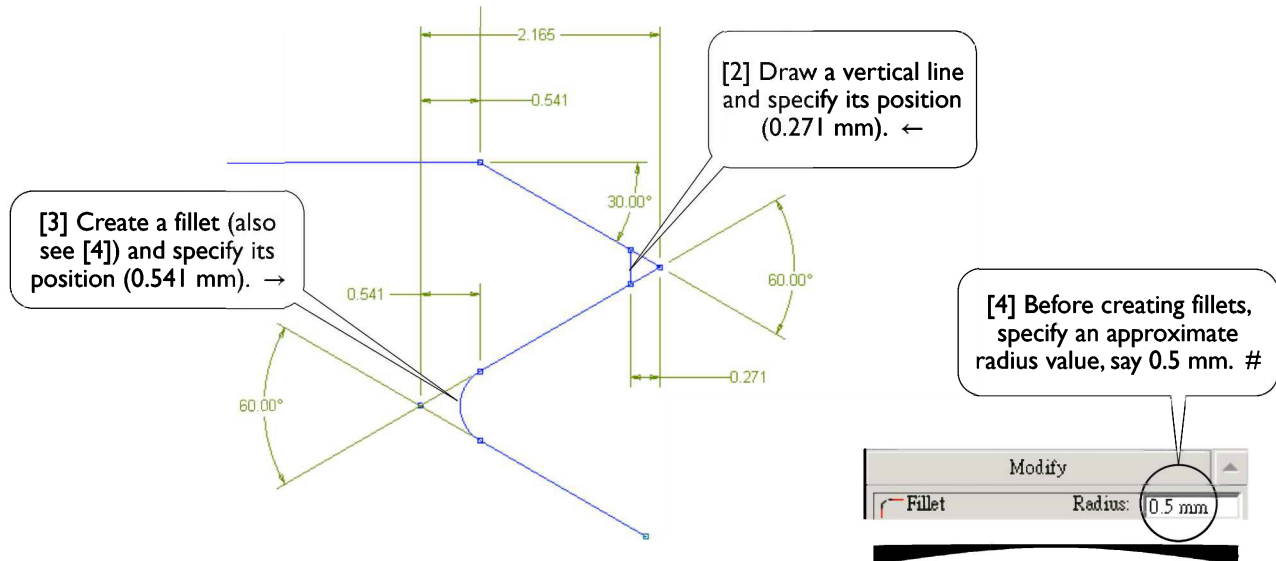
2.4.3 Draw a Polyline

[1] Draw a polyline of 3 segments [2-3] and specify the dimensions (30°, 60°, 60°, 0.541 mm, and 2.165 mm) as shown. To specify angle dimensions, please see **Angle**, 2.3.8[5], page 88. ↓

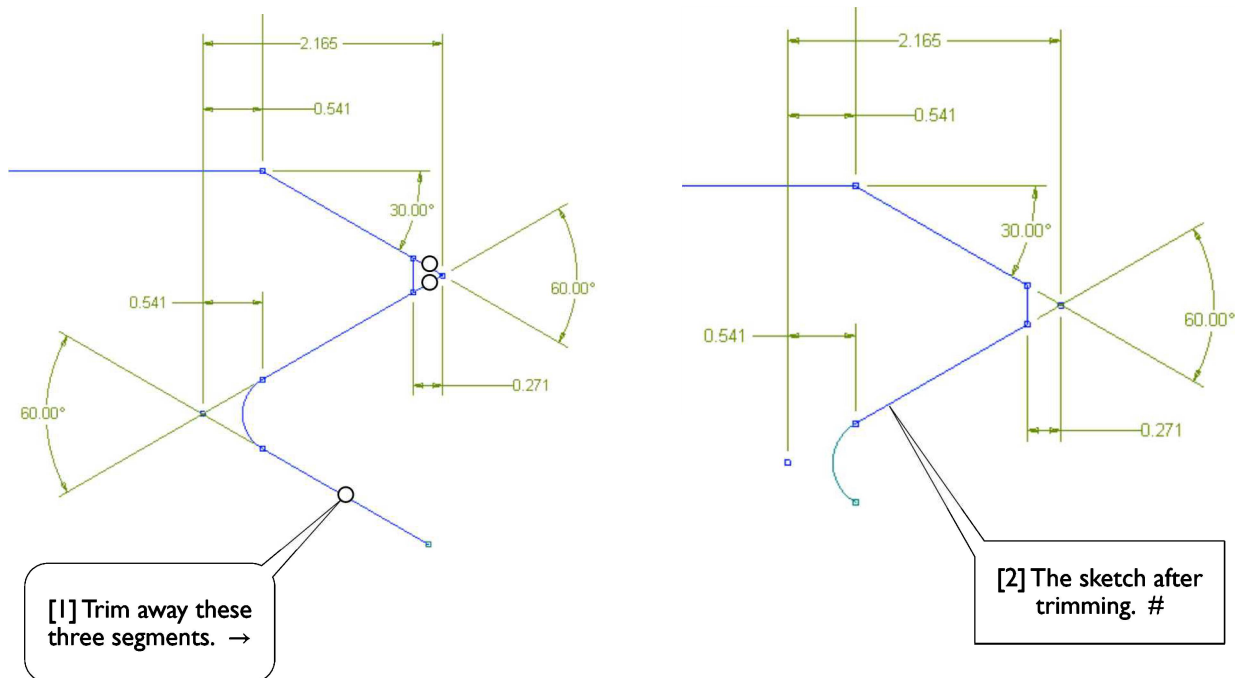


2.4.4 Draw a Line and a Fillet

[1] Draw a vertical line and specify its position [2].
Create a fillet and specify its position [3-4]. ↘



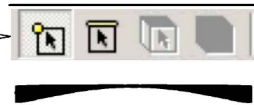
2.4.5 Trim Away Unwanted Segments



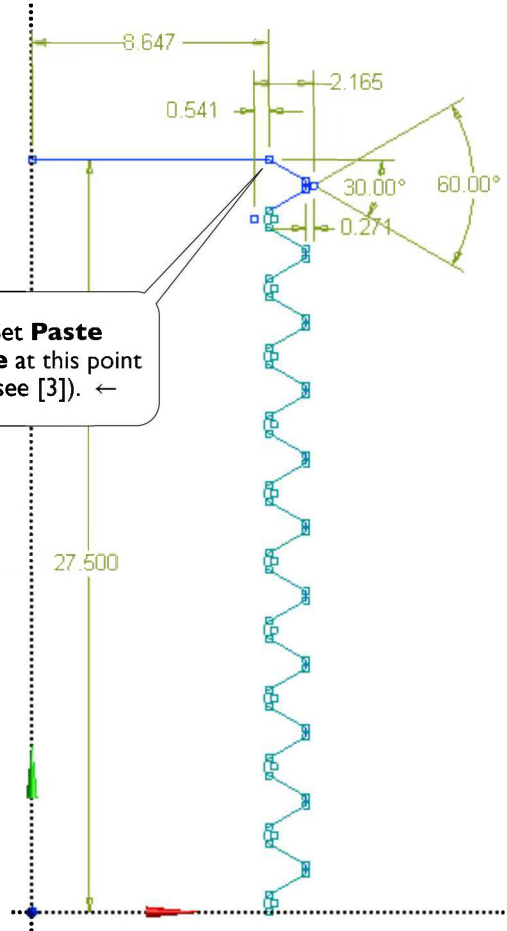
2.4.6 Replicate 10 Times

[1] Select the **Replicate** tool and select all segments except the horizontal line (4 segments in total), and replicate 10 times. Set the **Paste Handle** as shown in [2]. You may need to use **Selection Filter: Points** [3] (also see 2.2.8[7-8], page 74). ↓

[3] **Selection Filter: Points.** #



[2] Set **Paste Handle** at this point (also see [3]). ←



2.4.7 Complete the Sketch

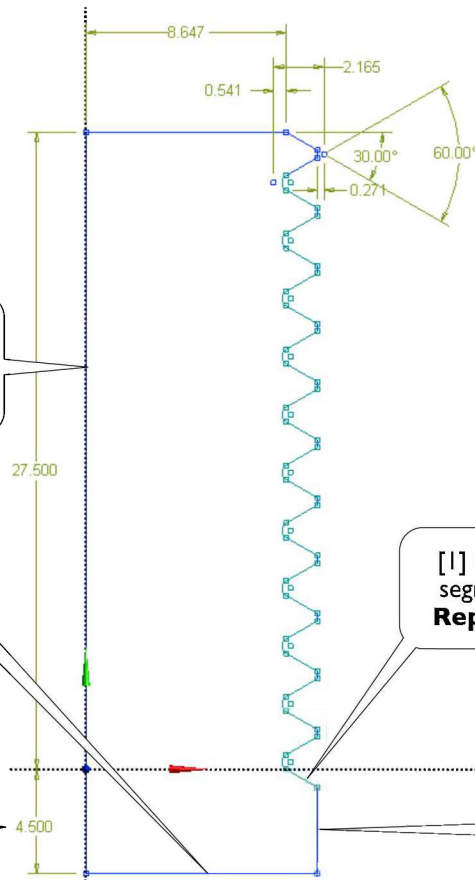
[2] Draw this vertical line, which passes through the origin. ↓

[5] Draw this horizontal line. #

[3] Specify this dimension (4.5 mm). →

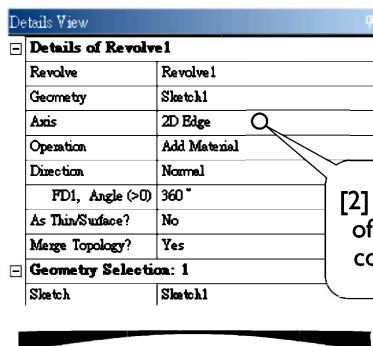
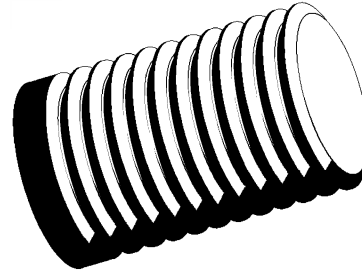
[1] Create this segment, using **Replicate**. ←

[4] Draw this vertical line. You may need to trim away extra length later after the next step. ←



2.4.8 Revolve to Create 3D Solid

[1] Click **Revolve** to generate a solid of revolution. Select the Y-axis as the axis of revolution [2]. Remember to click **Generate**. Save the project and exit Workbench. We will resume this project in Section 3.2. ↓



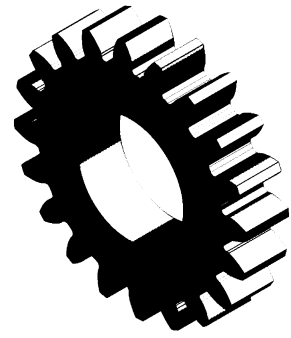
[2] Select the Y-axis as the **Axis** of revolution. (Make sure you correctly select the Y-axis.) #

References

1. Zahavi, E., *The Finite Element Method in Machine Design*, Prentice-Hall, 1992; Chapter 7. Threaded Fasteners.
2. Deutschman, A. D., Michels, W. J., and Wilson, C. E., *Machine Design: Theory and Practice*, Macmillan Publishing Co., Inc., 1975; Section 16-6. Standard Screw Threads.

Section 2.5

Spur Gears

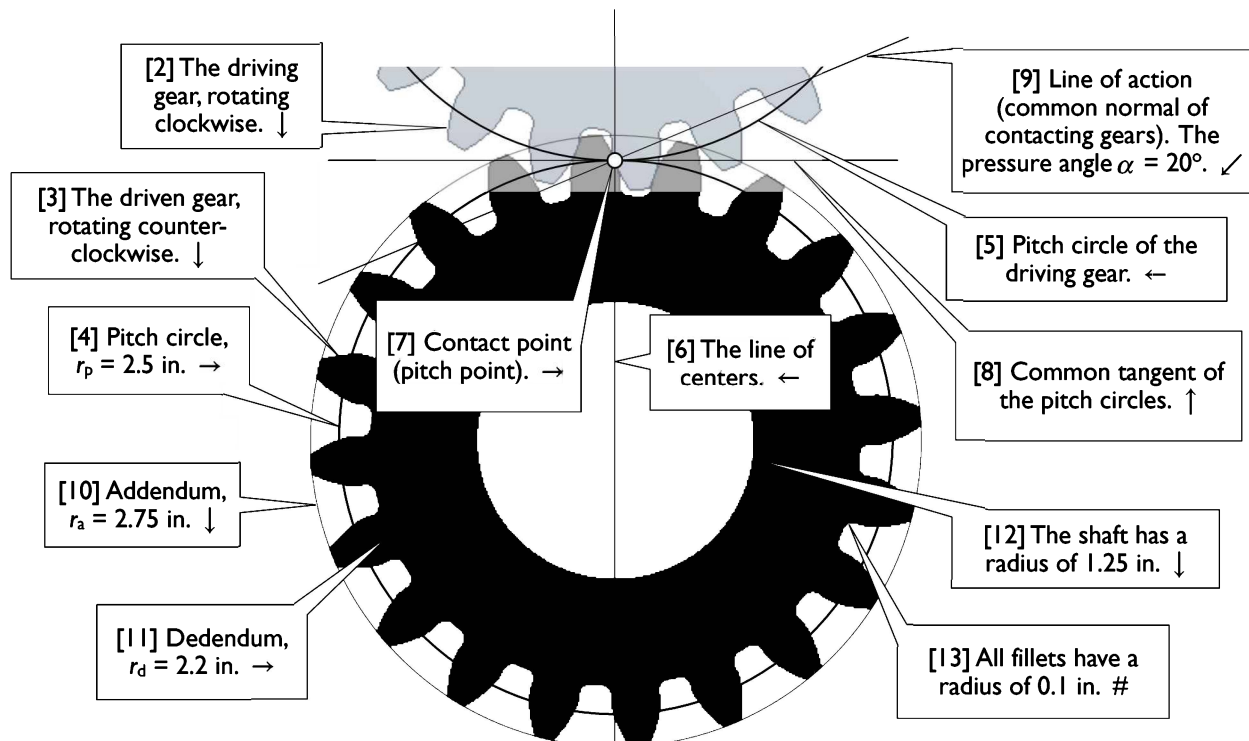


2.5.1 and 2.5.2 give the geometry of the spur gear used in this section. If you are not interested in these geometric details for now, you may skip them and jump directly to 2.5.3 (page 99).

2.5.1 About the Spur Gears^[Refs 1, 2]

[1] The figure below shows a pair of identical spur gears in mesh [2-5]. Spur gears have their teeth cut parallel to the axis of the shaft on which the gears are mounted, transmitting power between the parallel shafts. To maintain a constant angular velocity ratio, two meshing gears must satisfy a fundamental law of gearing: the shape of the teeth must be such that the common normal [9] at the point of contact between two teeth must always pass through a fixed point on the line of centers^[Ref 1] [6]. The contact point is called the *pitch point* [7].

The angle between the line of action [9] and the common tangent of the pitch circles [8] is known as the *pressure angle*. The spur gear is defined by its pitch radius ($r_p = 2.5$ in) [4], pressure angle ($\alpha = 20^\circ$) [9], and number of teeth ($N = 20$). The teeth are cut with a radius of addendum $r_a = 2.75$ in [10] and a radius of dedendum $r_d = 2.2$ in [11]. The shaft has a radius of 1.25 in [12]. All fillets have a radius of 0.1 in [13]. The thickness of the gear is 1.0 in. ✓



2.5.2 About Involute Curves^[Refs 1,2]

[1] To satisfy the fundamental law of gearing, gear profiles are usually cut to an *involute curve* [2], which may be constructed by wrapping a string (BA) around a *base circle* [3], and then tracing the path ($A-P-F$) of a point (A) on the string. Given the gear's pitch radius r_p and pressure angle α , we can calculate the coordinates of each point on the involute curve. ↘

[4] For example, let's calculate the polar coordinates (r, θ) of an arbitrary point A [5] on the involute curve. Note that BA and CP are tangent lines of the base circle, and F is a foot of perpendicular.

Since APF is an involute curve and $BCDEF$ is the base circle, by the definition of involute curve,

$$\overline{BA} = \overline{BCDEF} \quad (1)$$

$$\overline{CP} = \overline{CDEF} \quad (2)$$

In $\triangle OCP$,

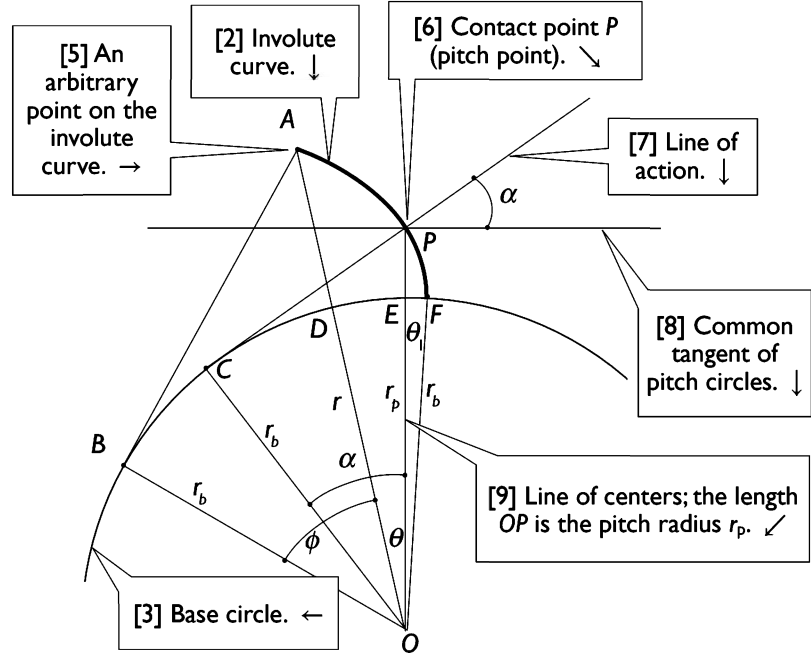
$$r_b = r_p \cos \alpha \quad (3)$$

In $\triangle OBA$,

$$r = \frac{r_b}{\cos \phi} \quad (4)$$

Or,

$$\phi = \cos^{-1} \frac{r_b}{r} \quad (5) \nearrow$$



[10] To calculate θ , we notice that

$$\widehat{DE} = \widehat{BCDEF} - \widehat{BCD} - \widehat{EF}$$

Dividing the equation with r_b and using Eq. (1),

$$\frac{\widehat{DE}}{r_b} = \frac{\widehat{BA}}{r_b} - \frac{\widehat{BCD}}{r_b} - \frac{\widehat{EF}}{r_b}$$

If radian is used, then the above equation can be written as

$$\theta = (\tan \phi) - \phi - \theta_1 \quad (6)$$

The last term θ_1 is the angle $\angle EOF$, which can be calculated by dividing Eq. (2) with r_b ,

$$\frac{\overline{CP}}{r_b} = \frac{\overline{CDEF}}{r_b}, \text{ or } \tan \alpha = \alpha + \theta_1, \text{ or}$$

$$\theta_1 = (\tan \alpha) - \alpha \quad (7)$$

We'll show how to calculate polar coordinates (r, θ) using Eqs. (3-7). The polar coordinates then can be easily transformed to rectangular coordinates, using O as origin and OP as y -axis,

$$x = -r \sin \theta, \quad y = r \cos \theta \quad (8) \leftarrow$$

Numerical Calculations of Coordinates

[1] In our case, the pitch radius $r_p = 2.5$ in, and pressure angle $\alpha = 20^\circ$; from Eqs. (3) and (7) respectively,

$$r_b = 2.5 \cos 20^\circ = 2.349232 \text{ in}$$

$$\theta_1 = \tan 20^\circ - \frac{20^\circ}{180^\circ} \pi = 0.01490438 \text{ (rad)}$$

The table below lists the calculated coordinates. The values in the first column (r) are chosen such that, except the pitch point ($r = 2.5$ in), the intermediate points are at the quarter points between r_b ($r = 2.349232$ in) and r_o ($r = 2.75$ in). Also note that, when using Eqs. (6) and (7), radians are used as the unit of angles; in the table below, however, degrees are used. #

r in.	ϕ Eq. (5), degrees	θ Eq. (6), degrees	$x = -r \sin \theta$ in.	$y = r \cos \theta$ in.
2.349232	0.000000	-0.853958	-0.03501	2.3490
2.449424	16.444249	-0.387049	-0.01655	2.4494
2.500000	20.000000	0.000000	0.00000	2.5000
2.549616	22.867481	0.442933	0.01971	2.5495
2.649808	27.555054	1.487291	0.06878	2.6489
2.750000	31.321258	2.690287	0.12908	2.7470

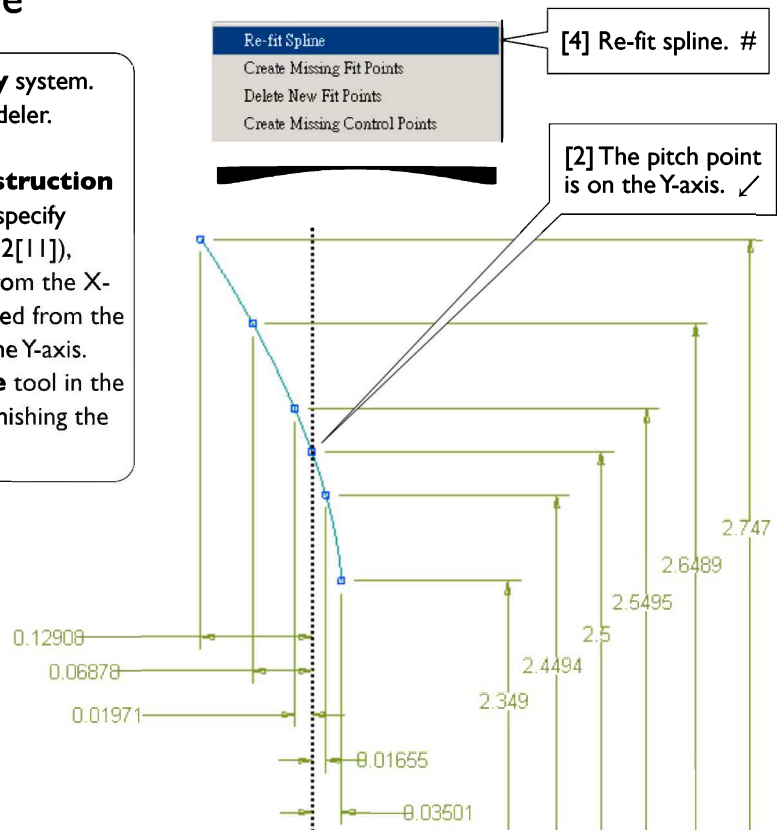
2.5.3 Draw an Involute Curve

[1] Launch Workbench. Create a **Geometry** system. Save the project as **Gear**. Start up DesignModeler. Select **Inch** as the length unit.

From the **Draw** toolbox, select the **Construction Point** tool, draw 6 points on **XYPlane** and specify dimensions as shown (also see the table in 2.5.2[1]), where the vertical dimensions are measured from the X-axis and the horizontal dimensions are measured from the Y-axis. The pitch point [2] is coincident with the Y-axis.

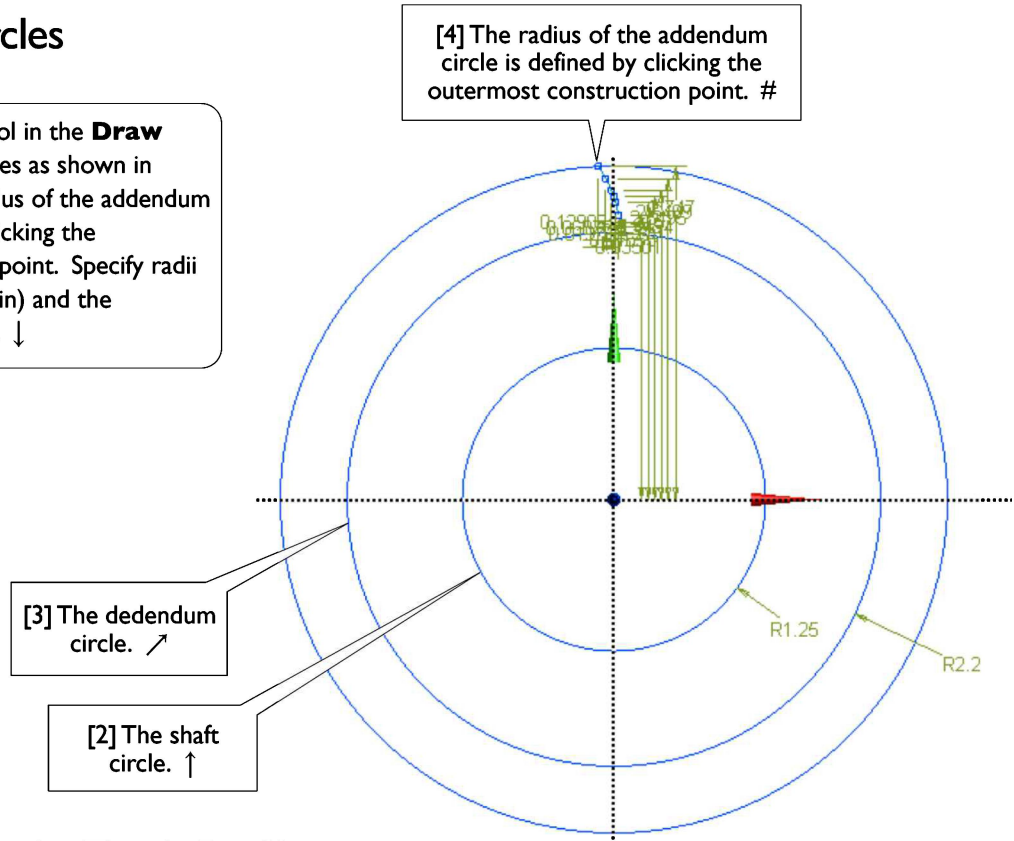
Connect these six points using the **Spline** tool in the **Draw** toolbox, leaving **Flexible** option on, finishing the spline with **Open End**. →

[3] It is equally good that you draw the spline by using the **Spline** tool directly without first creating construction points. To do so, at the end of the **Spline** tool, select **Open End with Fit Points** from the context menu. After dimensioning each point, use the **Spline Edit** tool to edit the spline and select **Re-fit Spline** [4] from the context menu to smooth out the spline. ↗



2.5.4 Draw Circles

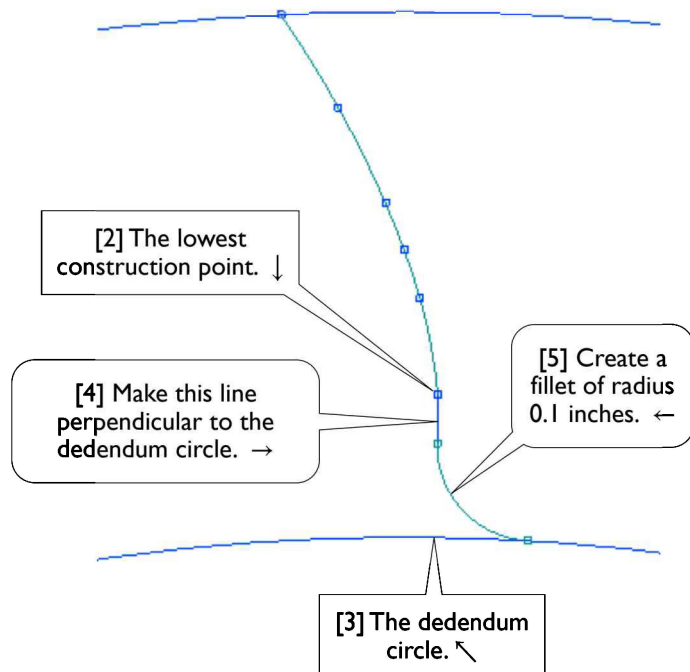
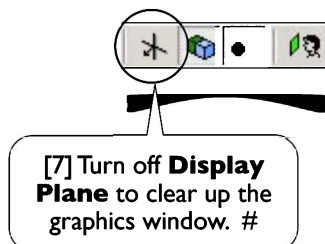
[1] Using the **Circle** tool in the **Draw** toolbox, draw three circles as shown in [2-4]. Note that the radius of the addendum circle [4] is defined by clicking the outermost construction point. Specify radii for the shaft circle (1.25 in) and the dedendum circle (2.2 in). ↓



2.5.5 Complete the Tooth Profile

[1] Draw a line from the lowest construction point [2] to the dedendum circle [3], and make it perpendicular to the dedendum circle by using the **Perpendicular** tool in the **Constraints** toolbox [4]. When drawing the line, avoid a **V** auto-constraint (since this line is NOT vertical). Create a fillet of radius 0.1 inches as shown in [5]. This completes the profile of a tooth. →

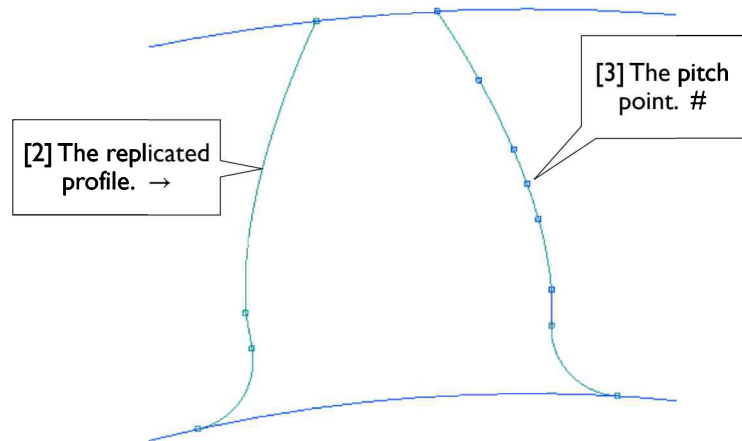
[6] Sometimes, turning off **Display Plane** may be helpful when working on the graphics window [7]. In this case, all the dimensions referring to the plane axes disappear. ↓



2.5.6 Replicate the Tooth Profile

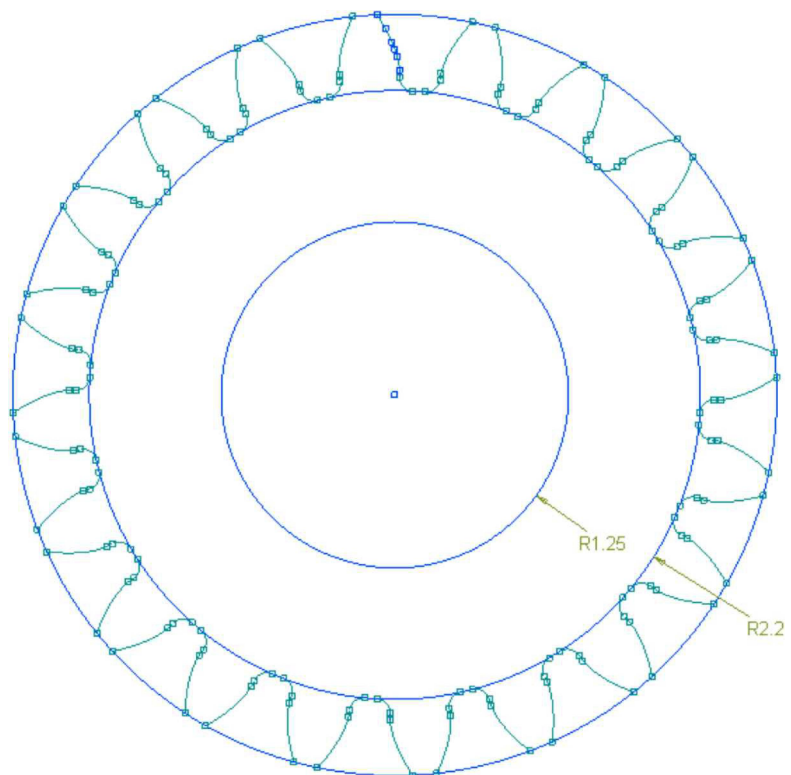
[1] Select the **Replicate** tool, type 9 (degrees) for **r**. Select the tooth profile (3 segments in total), **End/Use Plane Origin as Handle, Flip Horizontal, Rotate by r**, and **Paste at Plane Origin** [2]. End the **Replicate** tool by pressing **ESC**.

Note that the gear has 20 teeth, each spanning 18 degrees. The angle between the two pitch points [3] is 9 degrees. →



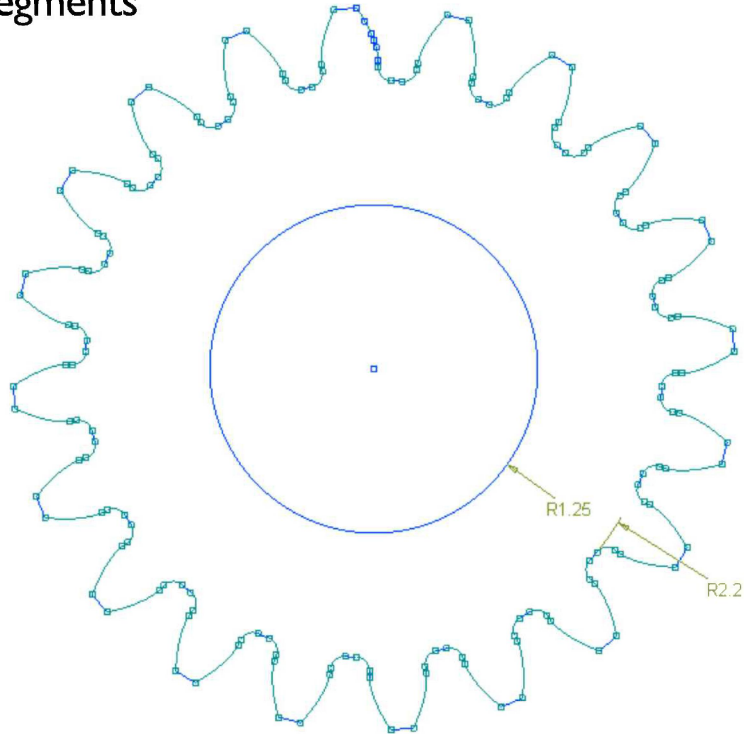
2.5.7 Replicate the Tooth 19 Times

[1] Click the **Replicate** tool again, type 18 (degrees) for **r**. Select both left and right profiles (6 segments in total), **End/Use Plane Origin as Handle, Rotate by r**, and **Paste at Plane Origin**. Repeat the last two steps (rotate and paste) until completing a full circle (20 teeth in total). #



2.5.8 Trim Away Unwanted Segments

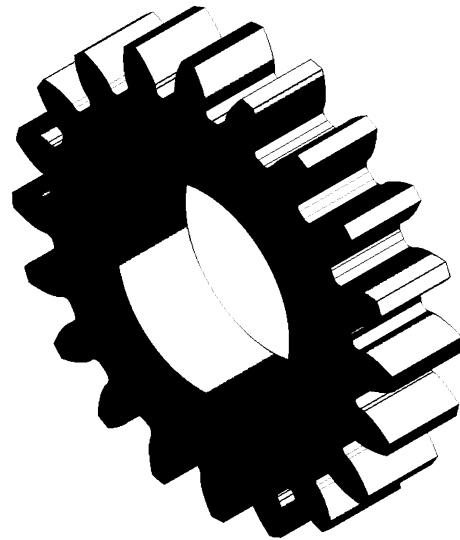
[1] Trim away unwanted segments in the addendum circle and the dedendum circle, as shown. #



2.5.9 Extrude to Create 3D Solid

[1] Extrude the sketch 1.0 inch to create a 3D solid. Save the project and exit from Workbench. We will resume this project in Section 3.4. ↓

[2] It is equally good that you create a single tooth (a 3D solid body) and then duplicate it by using **Create/Pattern** in the **Modeling** mode. In this exercise, however, we use **Replicate** in **Sketching** mode because our focus in this chapter is to practice sketching techniques. #

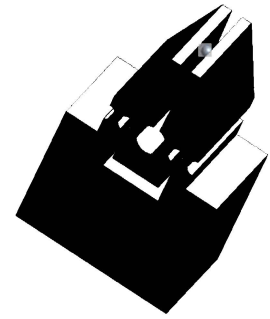


References

1. Deutschman, A. D., Michels, W. J., and Wilson, C. E., *Machine Design: Theory and Practice*, Macmillan Publishing Co., Inc., 1975; Chapter 10. Spur Gears.
2. Zahavi, E., *The Finite Element Method in Machine Design*, Prentice-Hall, 1992; Chapter 9. Spur Gears.

Section 2.6

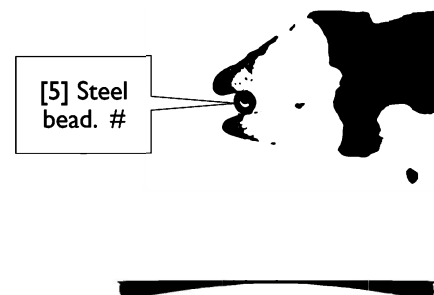
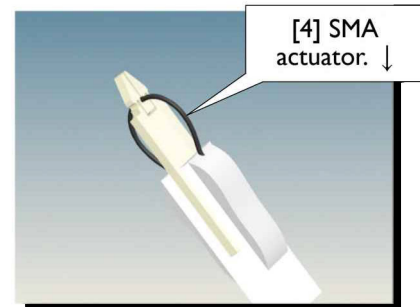
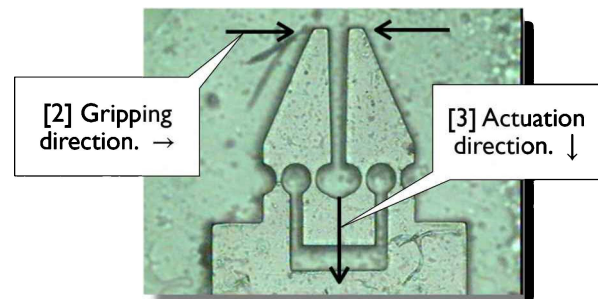
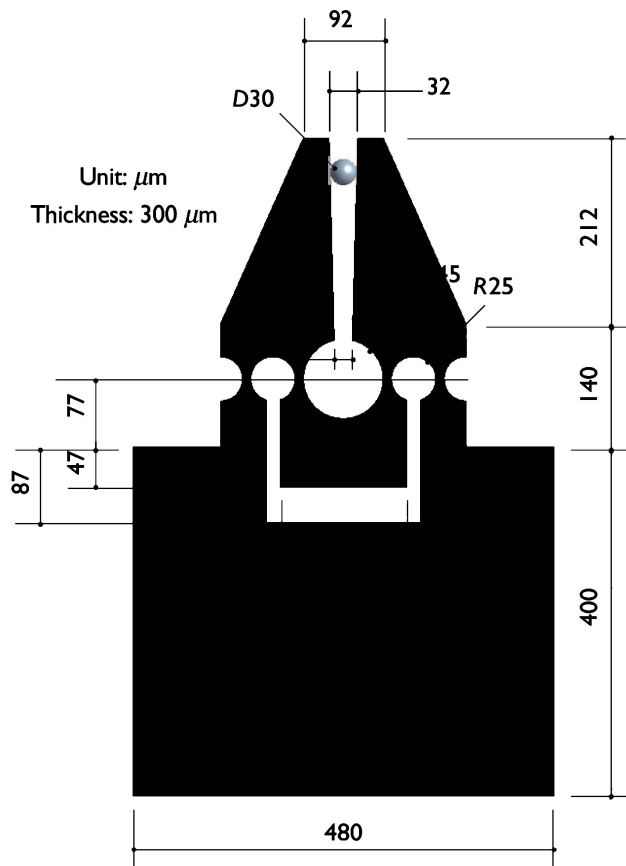
Microgripper



2.6.1 About the Microgripper^[Refs 1, 2]

[1] The microgripper is made of a rubber-like polymer material and actuated by a shape memory alloy (SMA) actuator [2-4]. The motion of the SMA is caused by temperature change, which is controlled by electric current. In the lab, the microgripper is tested by gripping a steel bead of a diameter of 30 micrometers [5].

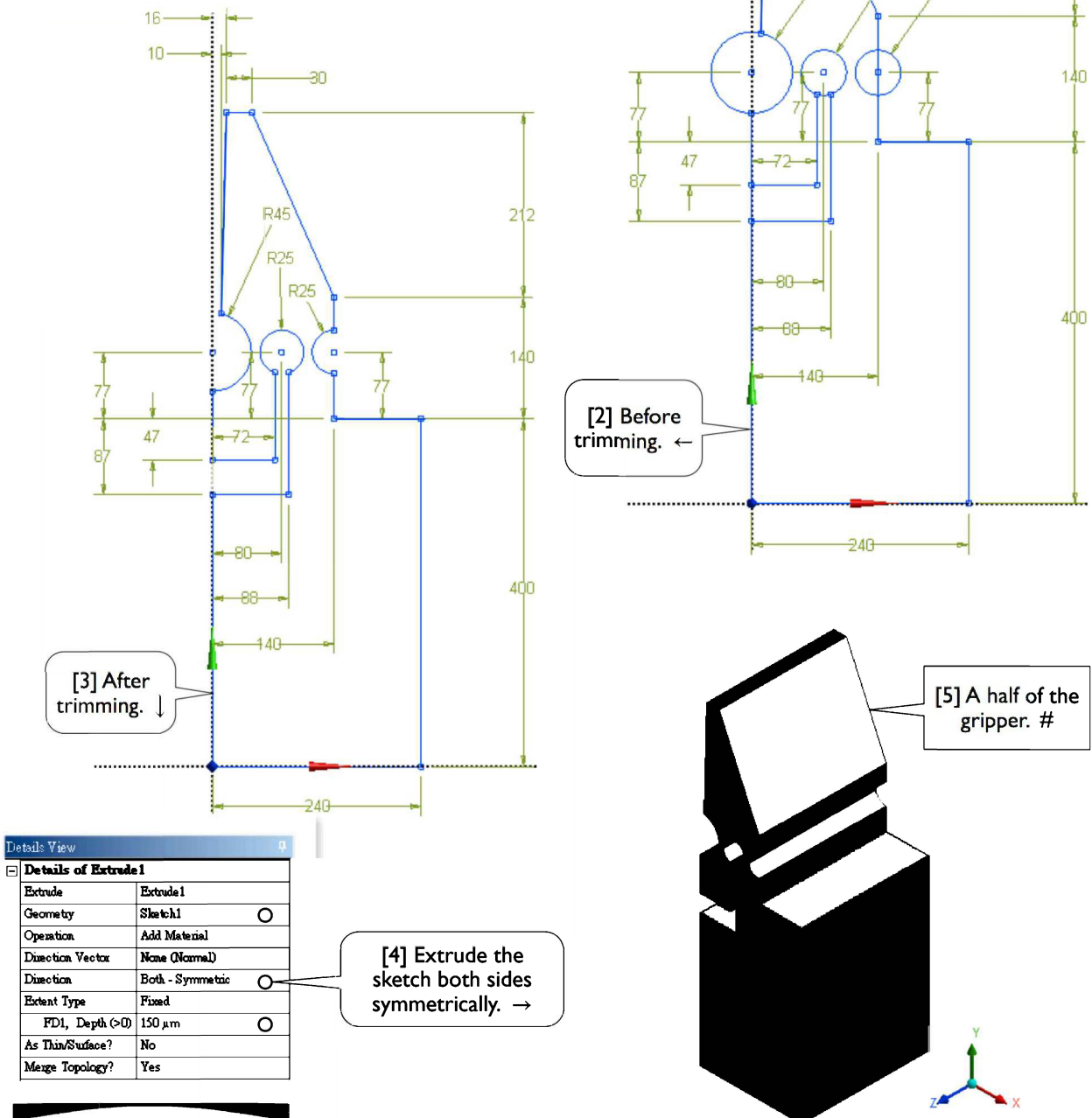
In this section, we will create a solid model for the microgripper. The model will be used for simulation in Section 13.3 to assess the gripping forces on the bead with an actuation force of the SMA actuator. ↓



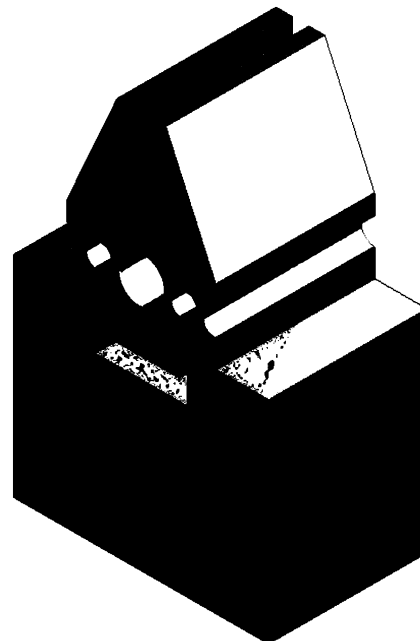
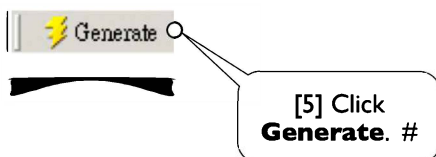
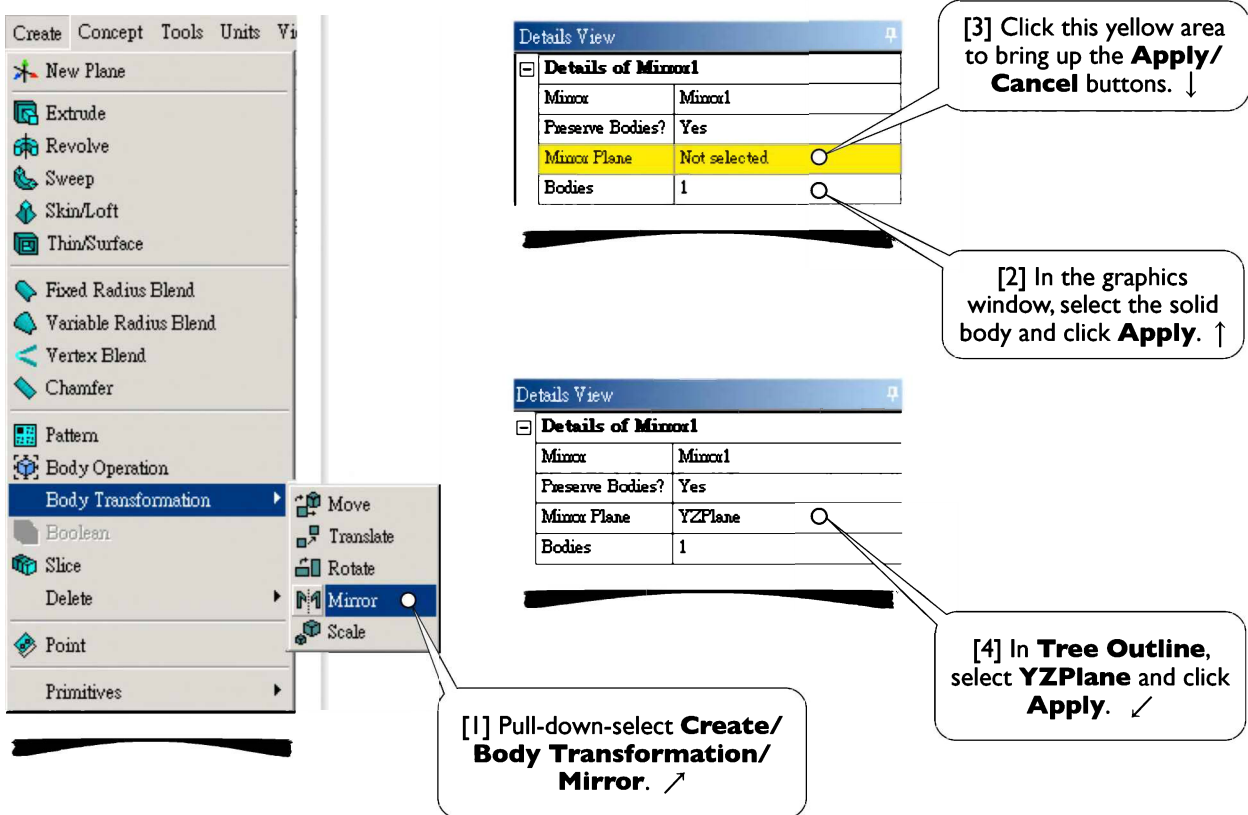
2.6.2 Create Half of the Gripper

[1] Launch Workbench. Create a **Geometry** system. Save the project as **Microgripper**. Start up DesignModeler. Select **Micrometer** as the length unit.

Draw a sketch on **XYPlane** as shown in [2]. Trim away unwanted segments [3]. Note that we drew only half of the model, due to the symmetry. Extrude the sketch 150 μm both sides symmetrically (the total depth is 300 μm) [4]. We now have a half of the gripper [5]. ↓



2.6.3 Mirror Copy the Solid Body

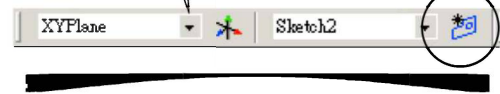


2.6.4 Create the Bead

[1] Create a new sketch on **XYPlane** as shown in [2-3] and draw a semicircle as shown in [4-7]. Revolve the sketch 360° about the Y-axis to create the bead [8]. Note that the two bodies are treated as two parts [9]. Rename the two bodies as **Gripper** and **Bead** respectively [10]. →

[2] Select **XYPlane**. →

[3] Click **New Sketch**. ✓



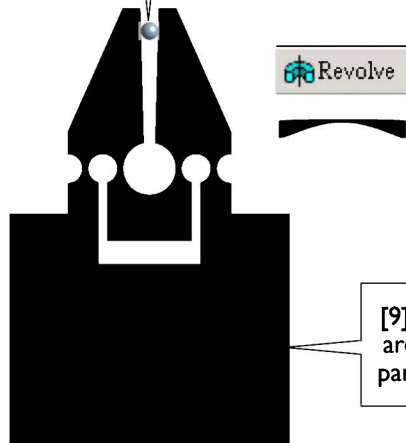
[4] The semicircle can be created by creating a full circle and then trimming it using the axis. ↓

[5] Close the sketch by drawing a vertical line. ✓

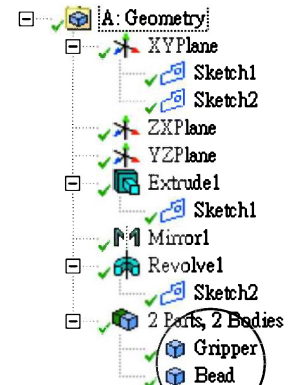
[8] **Revolve** the sketch about the Y-axis to create a sphere. ↓

[7] Impose a **Tangent** constraint between the semicircle and the sloping line. ✓

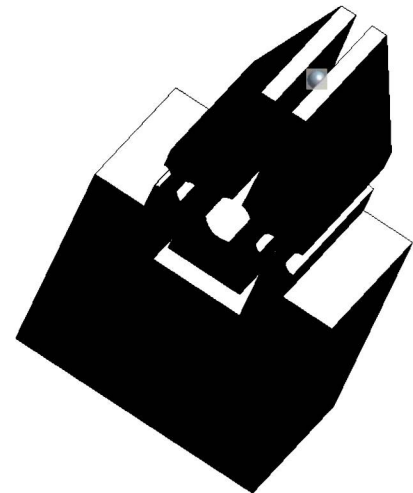
[6] Specify the radius (15 μm). ↑



[9] The two bodies are treated as two parts (see [10]). ✓



[10] Right-click to rename each body like this. ✓



Wrap Up

[11] Close DesignModeler, save the project and exit Workbench. We will resume this project in Section 13.3. #

References

1. Chang, R. J., Lin, Y. C., Shiu, C. C., and Hsieh, Y. T., "Development of SMA-Actuated Microgripper in Micro Assembly Applications," IECON, IEEE, Taiwan, 2007.
2. Shih, P. W., *Applications of SMA on Driving Micro-gripper*, MS Thesis, NCKU, ME, Taiwan, 2005.

Section 2.7

Review

2.7.1 Keywords

Choose a letter for each keyword, from the list of descriptions

- | | |
|----------------------------|----------------------------|
| 1. () Auto Constraints | 8. () Object |
| 2. () Branch | 9. () Paste Handle |
| 3. () Constraint Status | 10. () Sketching Mode |
| 4. () Context Menu | 11. () Sketching Plane |
| 5. () Edge | 12. () Sketch |
| 6. () Modeling Mode | 13. () Selection Filter |
| 7. () Model Tree | |

Answers:

1. (J) 2. (G) 3. (M) 4. (I) 5. (D) 6. (B) 7. (F) 8. (H)
 9. (L) 10. (A) 11. (C) 12. (E) 13. (K)

List of Descriptions

- (A) An environment under DesignModeler; its function is to draw sketches on a plane.
- (B) An environment under DesignModeler; its function is to create 3D or 2D bodies.
- (C) The plane on which a sketch is created. Each sketch must be associated with a plane; each plane may have multiple sketches on it. Usage of planes is not limited for storing sketches.
- (D) In **Sketching** mode, it may be a (straight) line or a curve. A curve may be a circle, ellipse, arc, or spline.
- (E) It consists of points and edges. Dimensions and constraints may be imposed on these entities.
- (F) It is the structured representation of a geometry and displayed on **Tree Outline** in **DesignModeler**. It consists of features and a part branch; their order is important. The parts are the only objects exported to **Mechanical**.

- (G) An object of a model tree and consists of one or more objects under itself.
- (H) A leaf or branch of a model tree.
- (I) The menu that pops up when you right-click your mouse. The contents of the menu depend on what you click.
- (J) While drawing in **Sketching** mode, by default, DesignModeler attempts to detect the user's intentions and tries to automatically impose constraints on points or edges. Detection is performed over entities on the active plane, not just active sketch. It can be switched on/off in the **Constraints** toolbox.
- (K) It filters one type of geometric entity. When it is turned on/off, the corresponding type of entity becomes selectable/unselectable. In **Sketching** mode, there are two selection filters, namely points and edges filters. Along with these two filters, face and body selection filters are available in **Modeling** mode.
- (L) A reference point used in a copy/paste operation. The point is defined during copying and will coincide with a specified location when pasting.
- (M) In **Sketching** mode, entities are color coded to indicate their constraint status: greenish-blue for under-constrained; blue and black for well constrained (i.e., fixed in the space); red for over-constrained; gray for inconsistent.

2.7.2 Additional Workbench Exercises

Create Geometric Models with Your Own Way

After so many exercises, you should be able to figure out many alternative ways of creating the geometric models in this chapter. Try to re-create the models in this chapter using your own way.

Chapter 3

2D Simulations

All the real-world bodies are 3D bodies; there are no such things as 2D bodies. Some problems, however, can be simplified and simulated in a 2D space. As an example, consider an axisymmetric body subject to axisymmetric loads, in which all the particles with the same radial and axial coordinates (R and Y) share the same behaviors regardless of their tangential coordinate (θ). Thus, we can eliminate the tangential coordinate and reduce the problem to a 2D (in R - Y space) problem. Other 2D cases include *plane stress* problems and *plane strain* problems, which will be defined in Section 3.3.

Reducing a problem to 2D has many advantages over a 3D approach, and you should always do it whenever possible. These advantages include (a) simpler to build the geometry, (b) better mesh quality, (c) less computing time, (d) easier display and analysis of the results. In short, the simulation model becomes smaller and easier to handle. If the problem's nature is indeed 2D, it would not introduce inaccuracy for the solution.

Purpose of This Chapter

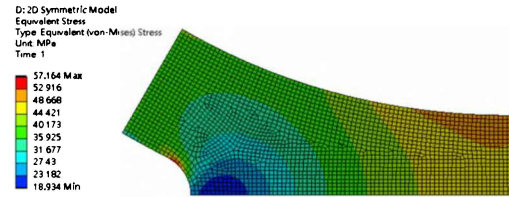
Since 2D simulations are usually easier to handle than 3D simulations, we start the learning of simulations by conducting 2D static structural simulations in this chapter, using some of the mechanical parts that we created in Chapter 2. This chapter also serves as a preliminary to 3D simulations, since most of the techniques and concepts in this chapter can be used in 3D simulations.

About Each Section

Sections 3.1 and 3.2 guide the students to perform a 2D simulation in a step-by-step fashion. Section 3.3 looks into more details and tries to provide what we are not able to cover in the first two sections. Section 3.4 provides an additional exercise. Problems in Sections 3.2 and 3.4 involve contact nonlinearities. In-depth discussion of contact nonlinearities will be postponed until Chapter 13. We introduce nonlinearities so early in this chapter to build motivation for learning nonlinear simulations in Chapters 13 and 14. Using a filleted bar subject to tension, Section 3.5 introduces some must-know concepts in finite element simulations, namely stress discontinuity, structural error, finite elements convergence, stress concentration, and stress singularity.

Section 3.1

Triangular Plate



3.1.1 About the Triangular Plate

[1] In this section, we will perform a 2D static structural simulation using the triangular plate created in Section 2.2. The plate is made of steel and used to withstand a tensile force of 20,000 N on each of its three side faces. The size of the side faces is $40 \times 10 \text{ mm}^2$; therefore, the applied tensile stress on the side faces is 50 MPa. The objective is to investigate the stresses in the plate.

We will model the problem as a 2D **plane stress** problem. Definition of the plane stress is given in 3.3.1, page 140; for now, what you need to know is that a thin plate subject to in-plane forces can be modeled as a plane stress problem.

There are two planes of symmetry in the model. In the first part of this section, we will analyze the full model without using the symmetries and then, in the second part of this section, reanalyze it by using the symmetries. #



3.1.2 Resume the Project Triplate

[1] Launch Workbench. →

[2] Open the project **Triplate**, which was saved in Section 2.2. ✓

[3] Right-click the title and select **Duplicate**. ✓

[4] Double-click the name and change to **3D Geometry**. →

[5] Double-click the name and change to **2D Geometry**. →

[6] Double-click this cell to open the geometric model. DesignModeler is brought forth. (Remember that the modeler is created with DesignModeler.) #

3.1.3 Delete the 3D Body and Create a 2D Body

[1] Right-click **Extrude1** and select **Delete**. Answer **Yes**. ↓

[2] Pull-down-select **Concept/Surfaces From Sketches**. ✓

[6] Click **Generate**. ↓

[3] Select **Sketch1**. ↓

[4] And click **Apply**. ↓

[5] We could type 10 (mm) here for the **Thickness**, but we decide to specify the thickness later in **Mechanical** (3.1.5[9], page 113). Leave it unchanged (0 mm) for now. ↑

[7] A surface body is created on **XYPlane**. The surface body will be treated as a 2D solid body (specified in 3.1.4[4-6], next page). ↗

[8] Close **DesignModeler**. #

3.1.4 Create Analysis System and Specify Analysis Type

[1] In **Toolbox**, under **Analysis Systems**, double-click **Static Structural** to create an analysis system. →

[2] Drag the **Geometry** cell of **2D Geometry**... →

[3] And drop to the **Geometry** cell of **Static Structural**. A link with a square end is created, meaning that the data are **SHARED** between the two cells. ↓

[4] Right-click **Geometry** and select **Properties**. ↓

[5] Select **2D** for **Analysis Type**, which (default to 3D) must be set before the geometry is exported to **Mechanical**. It cannot be changed after the geometry is brought to **Mechanical** (i.e., after the steps 3.1.5[1-2], next page). ↖

[6] Click to close **Properties**. ↓

Properties of Schematic B2: Geometry	
Property	Value
General	
Component ID	Geometry 2
Directory Name	Geom-1
Notes	
Notes	
Used Licenses	
Last Update Used Licenses	
Geometry Source	
Geometry File Name	C:\Users\ASUS\Documents\ANSYS\ANSYS2019\Sec03-1\Triplate_files\dp0\Geom-1\DM\Geom-1.agdb
Advanced Geometry Options	
Analysis Type	2D
Compare Parts On Update	No

Analysis Type

[7] For a 2D simulation, always set **Analysis Type** to 2D before the geometry is brought to **Mechanical**. If you make a mistake (entering **Mechanical** without setting the property), the only way to change **Analysis Type** is to clear up the entire database in **Mechanical**. To do so, quit **Mechanical**, right-click the **Model** cell, and select **Reset** from the context menu, as exemplified in 9.2.2[1], page 346. #

3.1.5 Start Up Mechanical and Set Up Geometry

[1] Double-click the **Model** cell to start up **Mechanical**. ↓

[2] A **Mechanical GUI** shows up. For the first-time use, click the **Geometry** tab if it is hidden. ↑

[3] Select **Geometry**. ↓

[4] Make sure **2D Behavior** is set to **Plane Stress**; this is the default 2D behavior. The term "plane stress" will be explained in 3.3.1, page 140. ↑

[5] By default, a **Ruler** is displayed in the bottom of the **Graphics Window**. To turn off the ruler, click the **Display** tab and select **Show/Ruler**. Throughout this book, we leave the ruler off (also see 2.1.4[2], page 61). ↘

[6] Click here and select **mm-kg-N-s** unit system. ↑

[7] Click **Zoom to Fit**. ↗

[8] Select **Surface Body**. ↙

[9] Type 10 (mm) for **Thickness** (also see 3.1.3[5], page 111). →

[10] Also note that the material assignment is **Structural Steel**. This is the default material. #

Project Schematic: Model (C4) contains Geometry, Materials, Coordinate Systems, Mesh, Static Structural (C5), Analysis Settings, Solution (C6), and Solution Information.

Static Structural Tree: 1 Static Structural, 2 Engineering Data, 3 Geometry, 4 Model, 5 Setup, 6 Solution, 7 Results.

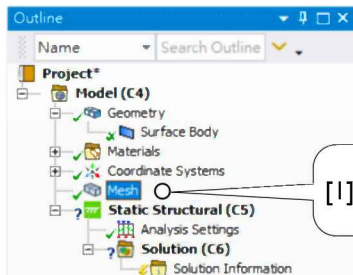
Graphics Window: Shows the Geometry tab, a ruler at the bottom, and a coordinate system.

Details of "Geometry": Definition (Source: C:\Users\ASUS\Documents\A..., Type: DesignModeler, Length Unit: Meters, Element Control: Program Controlled, 2D Behavior: Plane Stress, Display Style: Body Color), Bounding Box, Properties, Statistics, Update Options, Basic Geometry Options, Advanced Geometry Options.

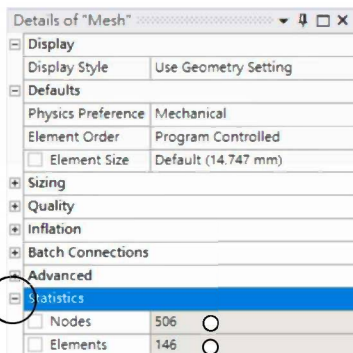
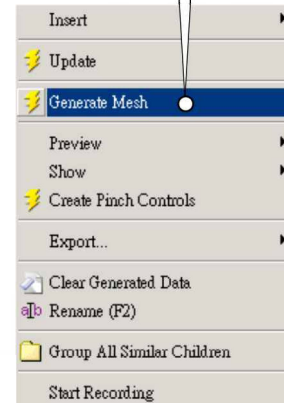
Details of "Surface Body": Graphics Properties, Definition (Suppressed: No, Stiffness Behavior: Flexible, Coordinate System: Default Coordinate System, Reference Temperature: By Environment, Thickness: 10 mm, Thickness Mode: Manual, Treatment: None), Material (Assignment: Structural Steel, Nonlinear Effects: Yes, Thermal Strain Effects: Yes), Bounding Box, Properties, Statistics, CAD Attributes, DMSheetThickness: 0.

Unit System: mm-kg-N-s

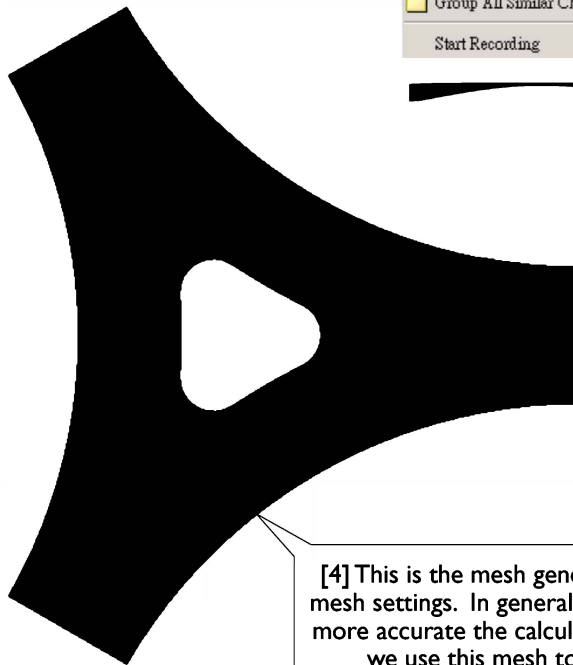
3.1.6 Generate Mesh



[2] And select **Generate Mesh**. ✓



[3] Expand **Statistics** to see the total numbers of nodes and elements. Your numbers may be slightly different from mine. ↘



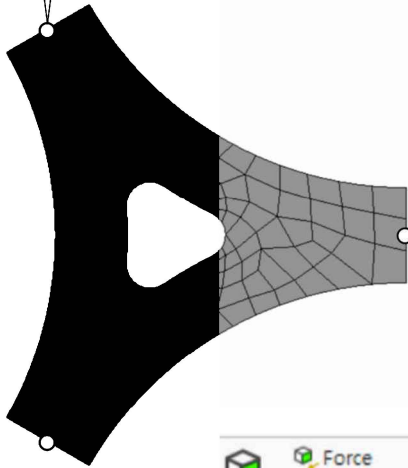
[4] This is the mesh generated with the default mesh settings. In general, the finer the mesh, the more accurate the calculated solution. For now, we use this mesh to obtain a solution. Remember, your mesh may be slightly different from mine. In 3.1.14 (pages 123-124), we'll show you a way to increase the mesh density. ↓

The steps in this subsection can be skipped.

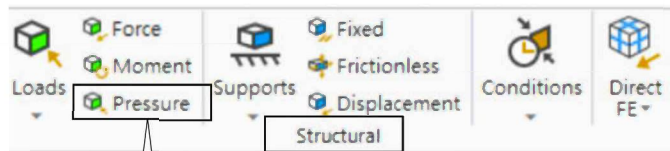
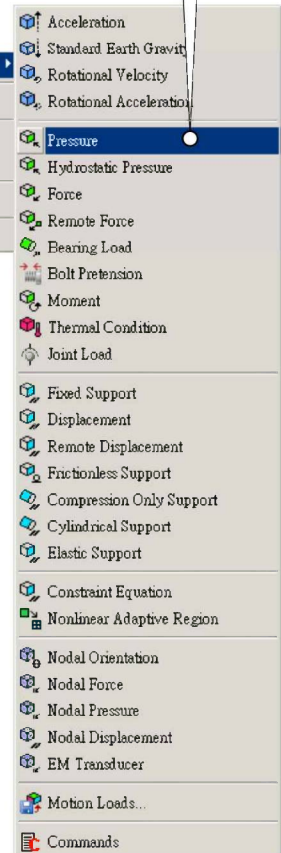
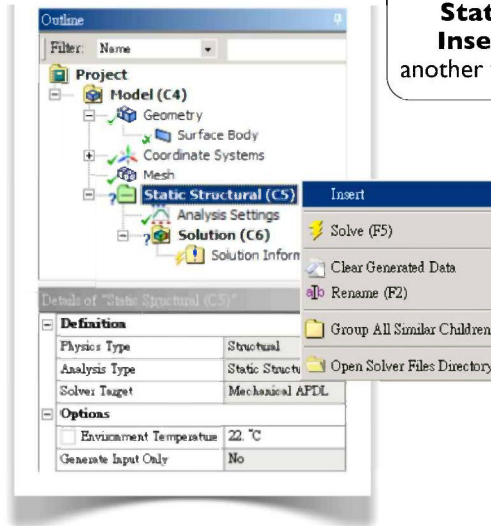
[5] When you issue **Solve** (3.1.8[3], page 116), Workbench automatically generates a mesh if one does not exist, as demonstrated in 3.1.12 (page 121). However, it is a good practice to preview the mesh before clicking **Solve**. We often mesh the model with default settings first, and then adjust mesh controls to improve the mesh. #

3.1.7 Specify Loads

[3] Hold down the **Control** key and select the three side edges. ↓



[1] Make sure nothing is selected in the **Graphics Window**. Right-click **Static Structural** and select **Insert/Pressure**. See [2] for another way to insert a **Pressure**. ✓



[2] Another way to insert a **Pressure** is to select **Environment/Structural/Pressure** from the toolbars. In ANSYS, the toolbars are called the **ribbon**. ↑

[4] And click **Apply**. ↓

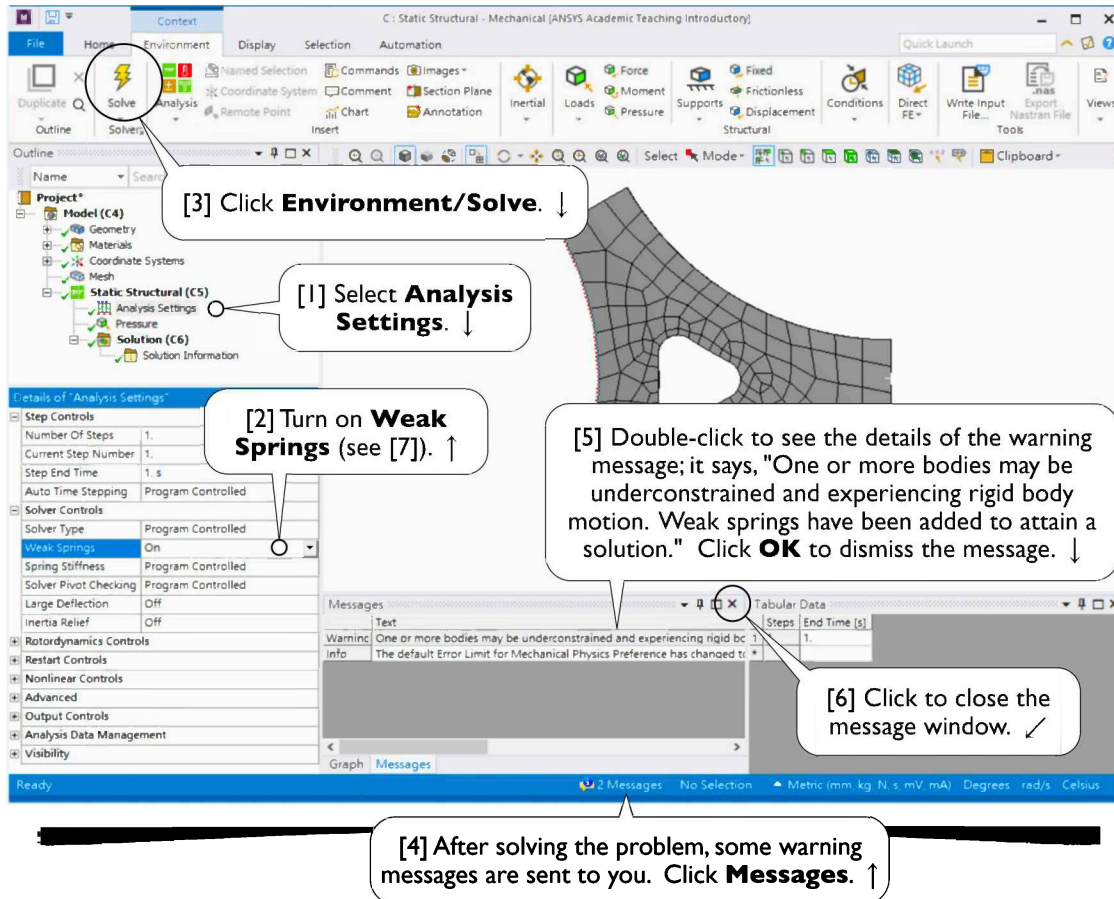
Details of "Pressure"	
Scope	
Scoping Method	Geometry Selection
Geometry	3 Edges
Definition	
Type	Pressure
Define By	Normal To
Applied By	Surface Effect
Magnitude	-50 MPa (capped)
Suppressed	No

[5] Type -50 (MPa) for **Magnitude**. A positive pressure pushes toward the body, while a negative pressure pulls away from the body. →

Context Menu versus Toolbar Menu

[6] Use of context menus [1] is instinctive: you select an object, right-click to open its context menu and choose a tool that operates on that object. Many frequently used tools are also available on the toolbars. Selecting a tool from the toolbars is usually more convenient than from a context menu. For example, with **Static Structural** highlighted, selecting **Environment/Structural/Pressure** [2] lets you insert a **Pressure** in the model tree. #

3.1.8 Solve the Model



About Weak Springs

[7] If a structure does not have enough supports, the structure is unstable and any non-zero external forces would cause the structure to move indefinitely; the motion is called a *rigid body motion*. An unstable structure still can achieve static equilibrium (unstable equilibrium) if the resultant external force is zero. This is what happens in our case.

Traditionally, in a static structural analysis, whenever a finite element program detects an unstable structure, it stops and reports an error message. This has been practiced for decades, even if the resultant external force is zero. Due to the nature of floating-point numerical processing in a digital computer, the external forces rarely sum up to zero. There is usually a small residual force left when summing up the forces. In other words, an unstable structure will undergo rigid body motion even if the resultant external force is theoretically zero.

In a Workbench's static structural analysis, if **Weak Springs** is turned on [2], **weak springs** will be added to an unstable structure, so it can withstand small external forces. They are called **weak springs** because the spring constants are small and negligible. With **weak springs**, rigid body motions are small but still perceivable in an unstable structure even when the external force is zero.

Anyway, it is a good practice to provide enough supports. In our case, you might set up a fixed support on one of the side faces and apply pressure of -50 MPa on the other two faces. That way, all deformations reported are relative to the fixed edge. A better way to model this case is to use the symmetry conditions of the structure. We will present the procedure in the second part of this section, starting from 3.1.11. #

3.1.9 Insert Result Objects

[1] Make sure nothing is selected in the **Graphics Window**. Right-click **Solution** in the model tree and select **Insert/Deformation/Total**.

[2] Right-click **Solution** again and select **Insert/Stress/Equivalent (von-Mises)**.

[3] Two result objects are inserted to the project tree, under the **Solution** branch.

[4] Click **Solution/Solve** (which is the same tool as that in 3.1.8[3], last page). This is to evaluate the result objects. #

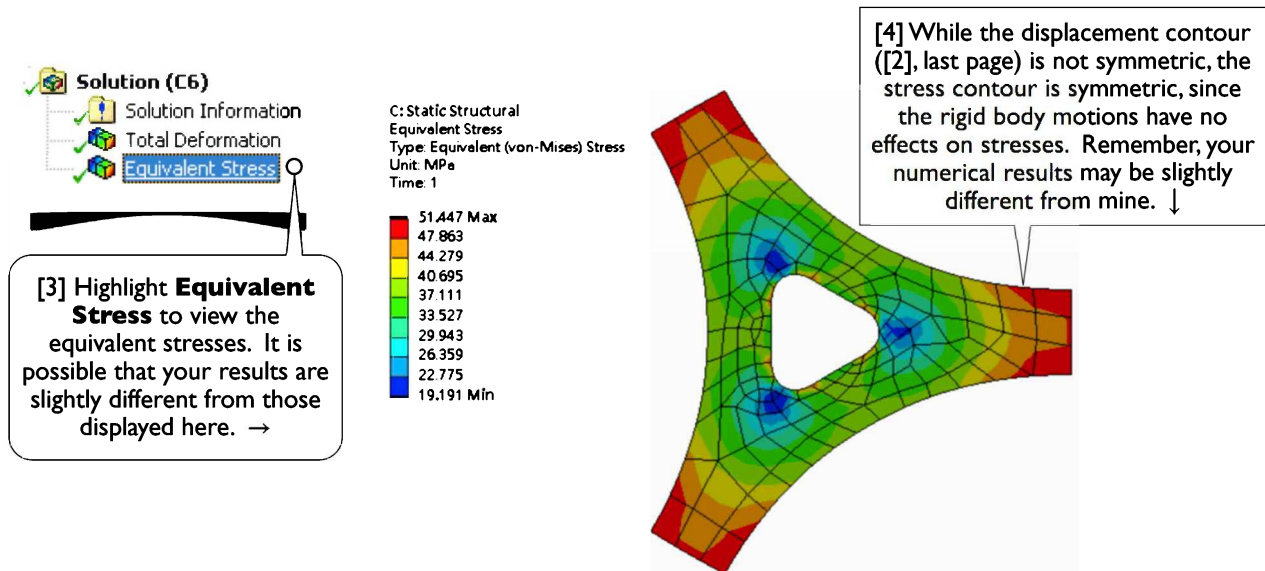
[2] Since the structure is not fully supported, the results include a small rigid body motion. That is why the displacement contour is not symmetric. Also note that your numerical results may be slightly different from mine. ↵

3.1.10 View the Results

[1] Highlight **Total Deformation** to view the deformations. It is possible that your results are slightly different from mine. →

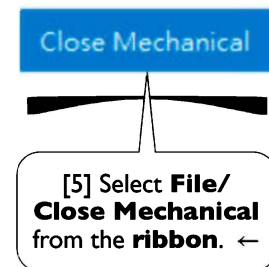
C: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1

0.021066 Max
0.01875
0.016433
0.014116
0.011799
0.0094822
0.0071653
0.0048485
0.0025316
0.00021477 Min



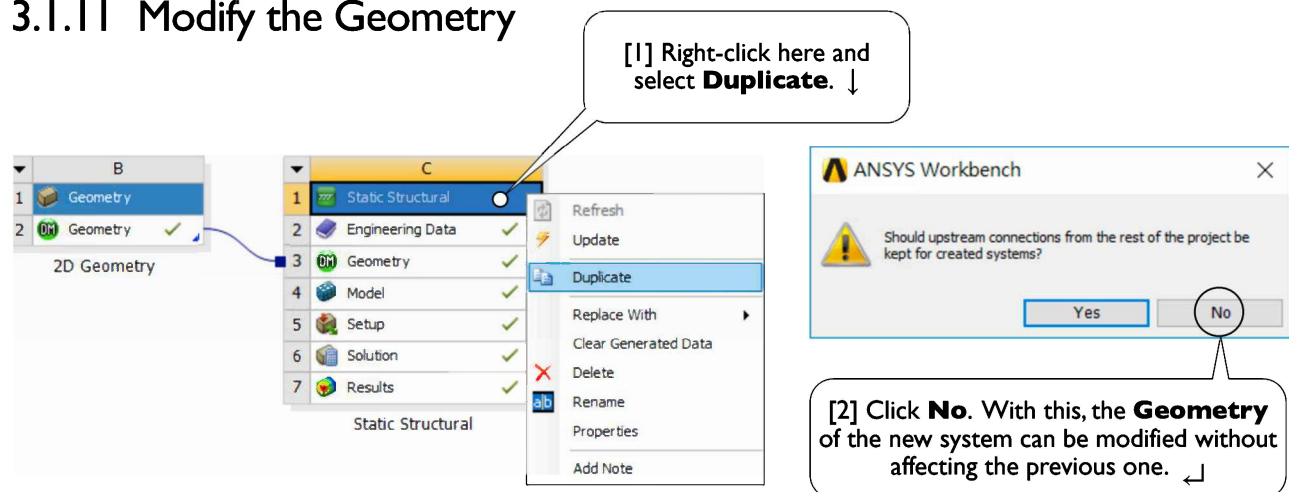
Symmetry of the Response

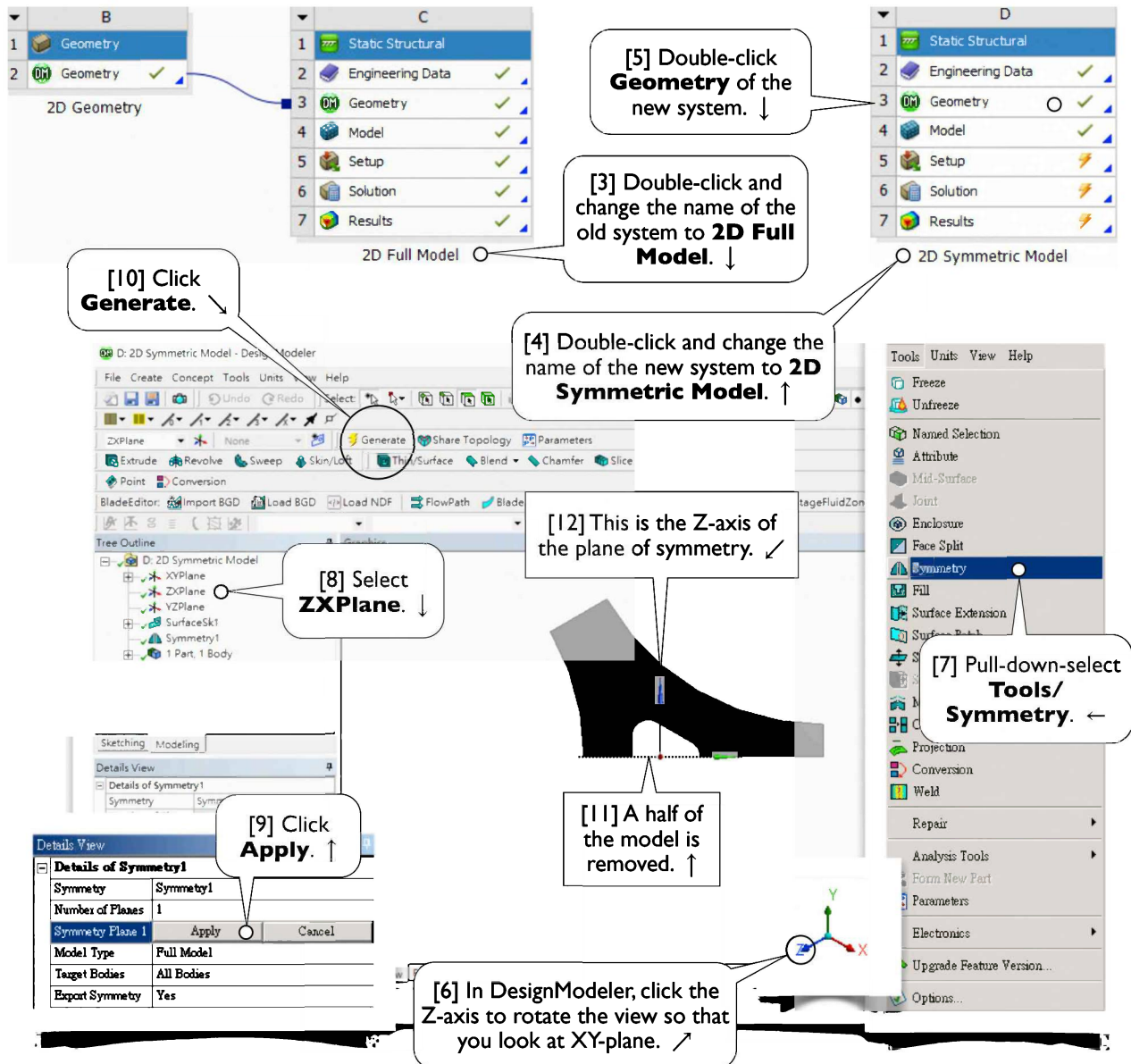
[6] While the stress contour [4] is symmetric, you may wonder why the displacement contour ([2], last page) is not symmetric. Theoretically, the displacements should be symmetric. However, because the structure is not fully supported, the results include a small rigid body motion, due to the nature of floating-point numerical processing. That is why the displacement contour is not symmetric [2]. A way to avoid this kind of numerical problem is to use symmetry conditions, as demonstrated below. #



SIMULATION OF SYMMETRIC MODEL

3.1.11 Modify the Geometry



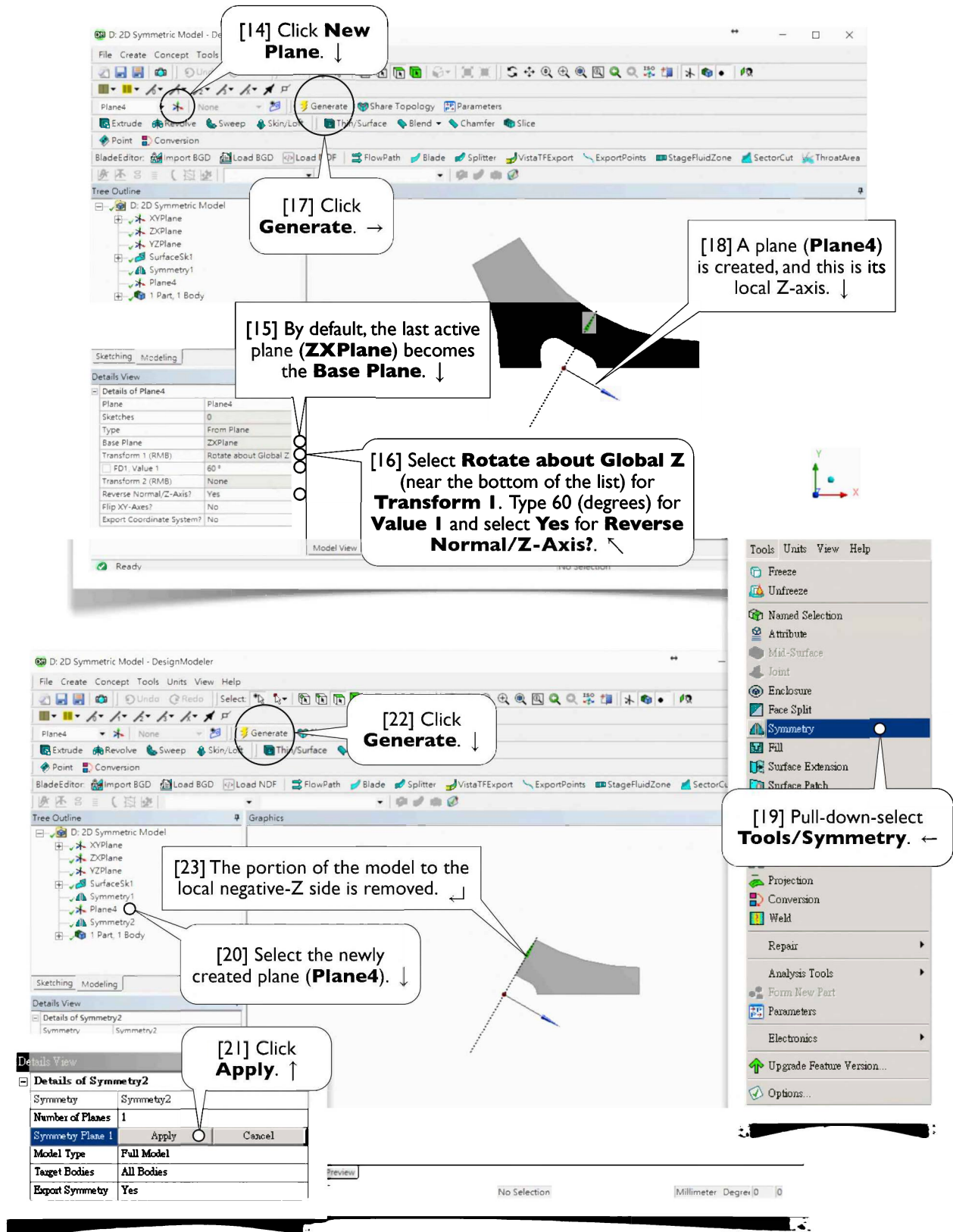


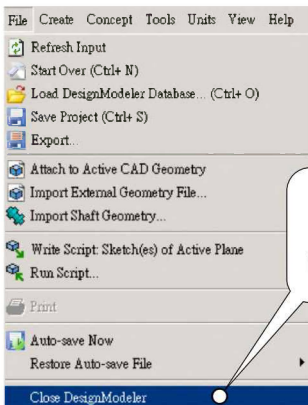
About Coordinate Systems

[13] There is a unique global coordinate system; its three axes are shown in the bottom-right corner of the graphics window [6]. Workbench uses R, G, and B colors to represent X-, Y-, and Z-axis respectively: red arrow for X-axis, green arrow for Y-axis, and blue arrow for Z-axis. In this book, we use upper-case (X, Y, Z) for both global and local coordinate systems, to be consistent with the notations used in Workbench.

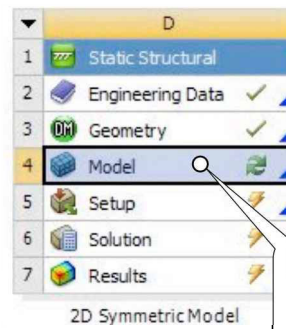
Each plane has its own local coordinate system, using the same color codes. Let's take **ZXPlane** as an example [12]. Its local XY-plane coincides with global ZX-plane, and the local Z-axis points upward. When we specify a plane as the plane of symmetry, the plane's local XY-plane is used to cut away the portion of the model on the local negative-Z side. The portion of the model on the local positive-Z side remains [11].

The triangular plate has another plane of symmetry. None of the default planes can be used as the plane of symmetry; we need to create one. This plane can be derived from rotating **ZXPlane** by 60 degrees. ↙





[24] Pull-down-select **File/Close DesignModeler.** →



[25] Double-click **Model** (of the **2D Symmetric Model**) to start up **Mechanical**. Answer **Yes** to update the geometry. ↓

Do I need to close DesignModeler when working on **Mechanical**, or vice versa?

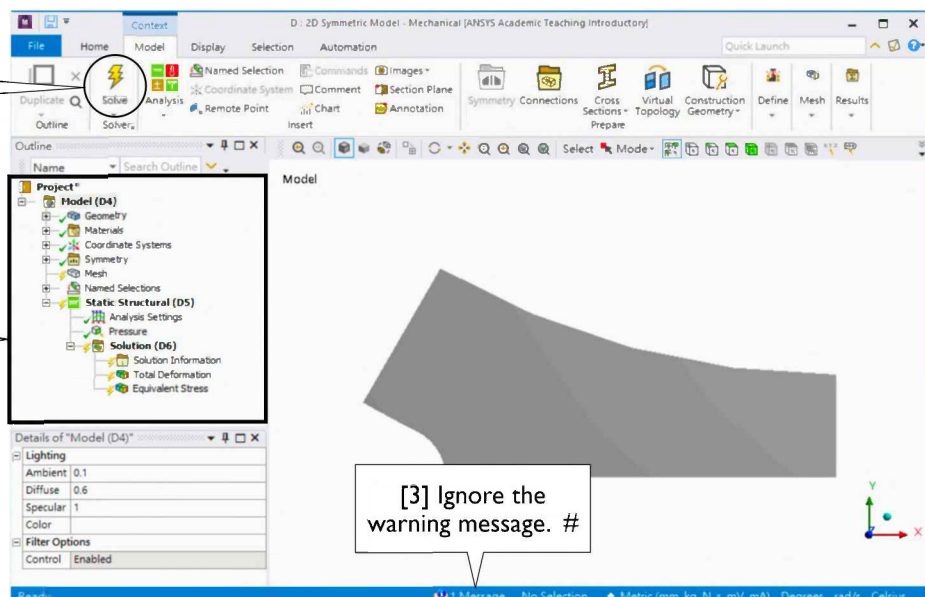
[26] When starting up, **Mechanical** automatically updates the geometry, including adding symmetry conditions on the model. I strongly suggest that the newcomers close one application while switching to another. That way, the update will be automatic; you won't mess up the work flow.

However, you don't have to close an application when switching to another application, but you have to "refresh" it by yourself. In **Project Schematic** you can right-click **Model** (of an analysis system) and select **Update Upstream Component**. Inside **Mechanical** you can right-click **Geometry** (of the project tree) and select **Update Geometry from Source**. Until you become an experienced user, I suggest that you always keep only one application open at a time to make life simpler. #

3.1.12 Solve the New Model

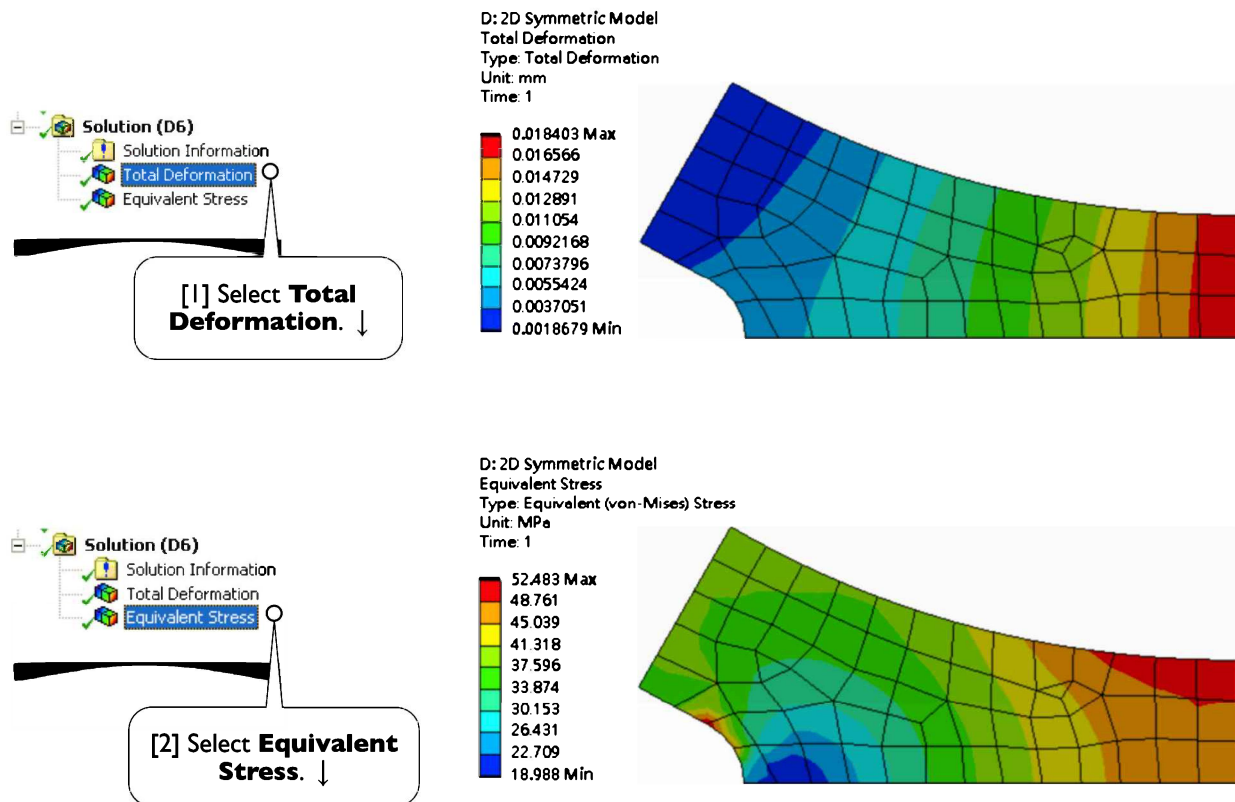
[2] Click **Model/ Solve.** #

[1] After importing the new geometry, many objects have "thunderbolts," indicating that the data are not up-to-date and need to be re-calculated. ↑



[3] Ignore the warning message. #

3.1.13 View the Results



Check Environment Conditions after Modifying Geometry

[3] In this case, you don't need to do anything before solving the new model. In most other cases, however, whenever you modify your geometry, you may need to redefine the environment conditions (supports, loads, etc.). As a good practice, always check your environment conditions each time you modify the geometry before solving it.

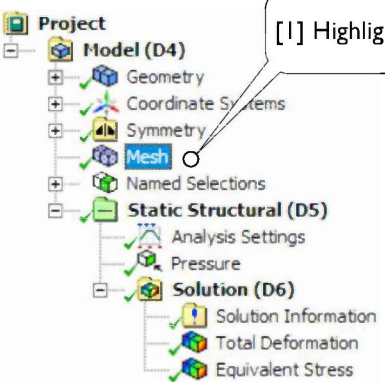
Why Different Numerical Results?

Your numerical results may be slightly different from mine. This is one of the characteristics of the finite element methods: different meshes end up with different results. In general, the finer the mesh, the more accurate the results. The question is how fine should a mesh be, to achieve enough accuracy? We will start to discuss this must-know concept in Section 3.5. The discussion will be extended to 3D cases in Section 9.3.

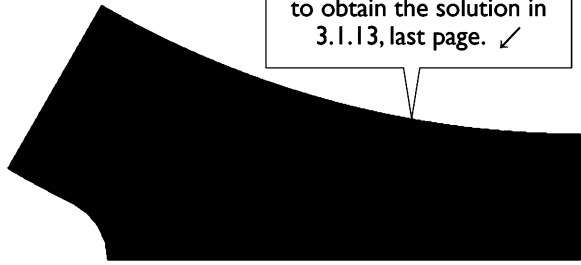
Some students may be puzzled about why they obtained a mesh different from the one in the book even though they followed EXACTLY the same steps as those in the book. The answer is that the students have no way to follow EXACTLY the same steps in the book. For example, a line in the book may be drawn from right to left while you drew it from left to right. It is possible that the direction of the line affects the meshing algorithm in the Workbench.

Limited differences in numerical values are normal, particularly when the mesh is coarse. As the mesh becomes finer, the solution will converge to a theoretical value, independent of mesh variations. This kind of puzzle would disappear. #

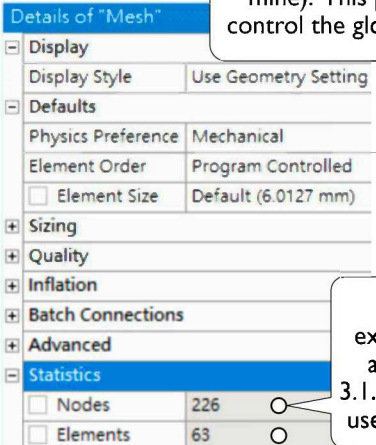
3.1.14 Improve Solution Accuracy




[1] Highlight **Mesh**. →



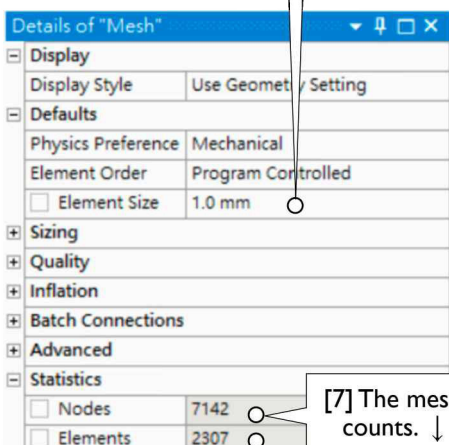
[2] This is mesh we used to obtain the solution in 3.1.13, last page. ✓



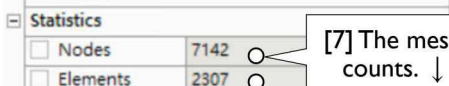
[3] The maximum element size determined by Workbench is about 6 mm (your number may be slightly different from mine). This parameter can be used to control the global fineness of the mesh. ↓



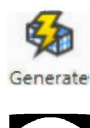
[4] Expand **Statistics** and examine the mesh counts. They are much fewer than those in 3.1.6[3] (page 114), meaning that it uses less computing resource. ↗




[5] Change **Element Size** to 1 mm. ↓



[7] The mesh counts. ↓



[6] From the **ribbon**, select **Mesh/Generate**. ↑



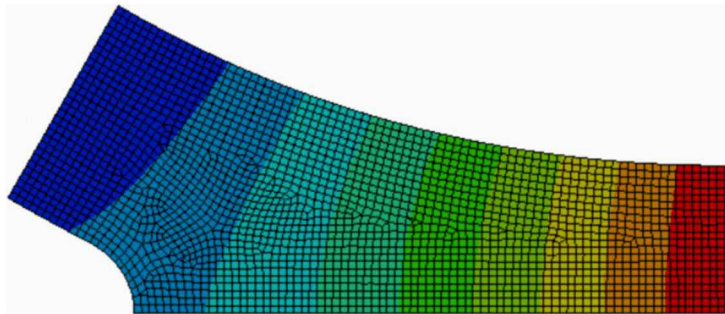
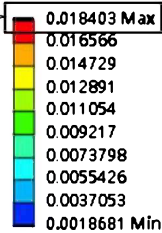
[8] The mesh is much finer than that before. ↙

[9] Click **Solve** to solve the model. Ignore the warning message. ←



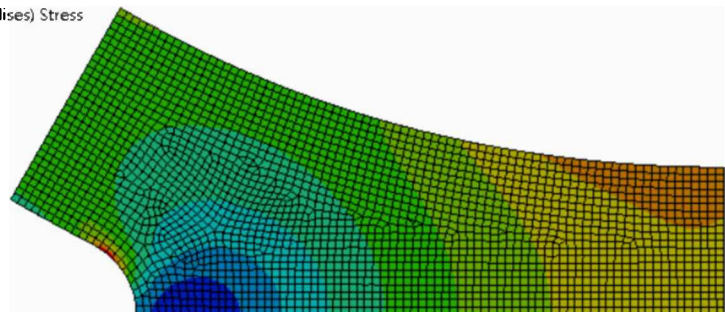
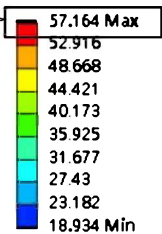
[10] Select **Total Deformation**. The displacements are almost the same as those in 3.1.13[1] (page 122). The implication is that, for displacement solution, the mesh shown in [2] is as adequate as that in [8]. ↓

D: 2D Symmetric Model
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1



[11] Select **Equivalent Stress**. The stresses are quite different from those in 3.1.13[2] (page 122). The implication is that, for stress solution, the mesh shown in [2] is not as adequate as that in [8]. ↓

D: 2D Symmetric Model
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1



Accuracy of Solution

[12] Here, we've demonstrated an important behavior of finite element simulations: the finer the mesh, the more accurate the calculated solution. The big question is: how fine is adequate? There is no easy answer for this question. A comprehensive way to assure an adequate mesh is to conduct a "convergence study," demonstrated in PART C of Section 3.5 (pages 161-163).

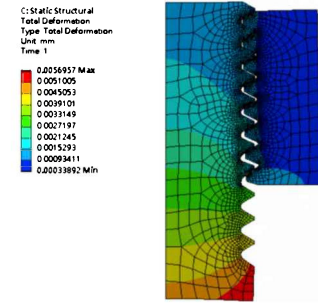
Another important concept is that the element sizes usually need not be uniform; an example is shown in 3.2.8[18], page 135. We usually require finer mesh at an area in which the change of stress is more dramatic, i.e., at an area that has larger stress gradient. Of course, you need lots of engineering experiences to cultivate a sense of "stress gradient." In the next section, we will demonstrate the mesh control at local areas. ↓

Wrap Up

[13] Close **Mechanical** (select **File/Close Mechanical**). Save the project and exit Workbench. #

Section 3.2

Threaded Bolt-and-Nut



3.2.1 About the Threaded Bolt-and-Nut

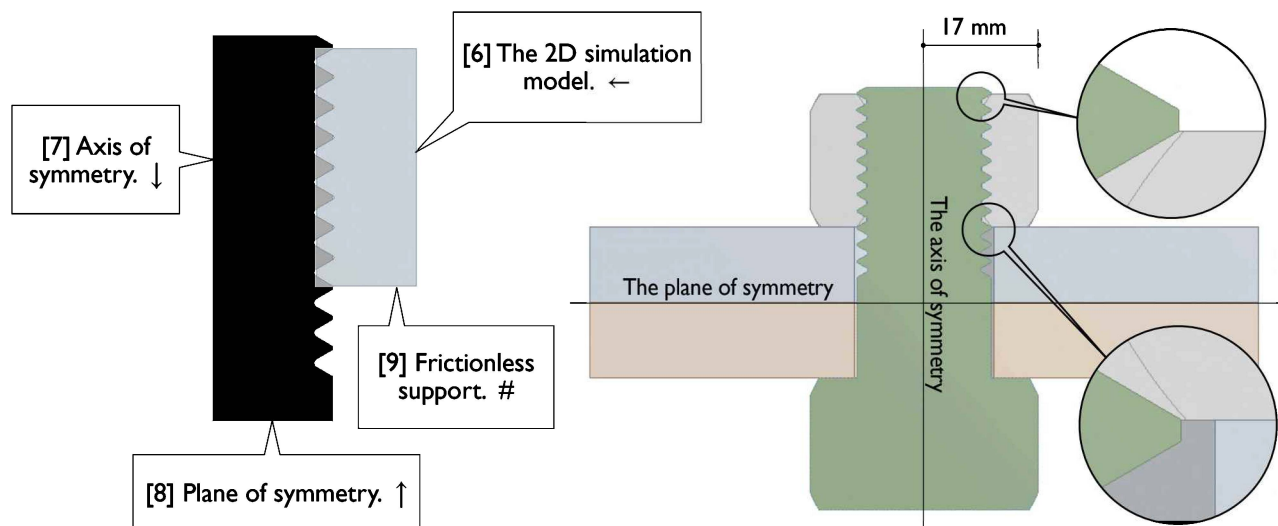
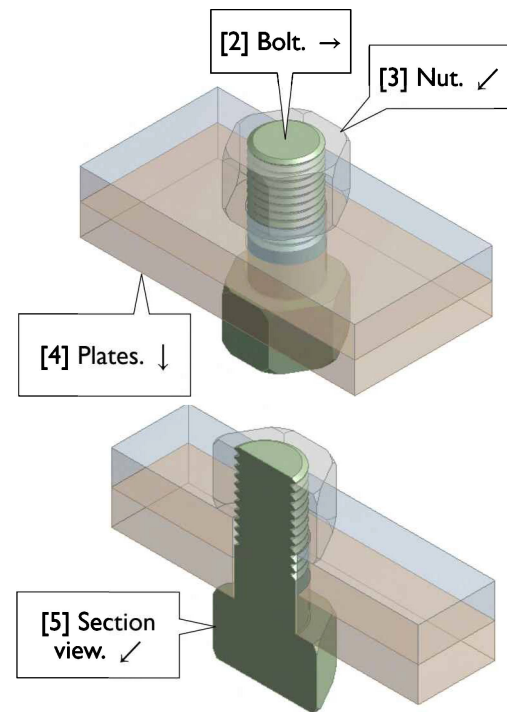
[1] The threaded bolt created in Section 2.4 is a portion of a bolt-nut-plate assembly [2-5]. The bolt is preloaded with a tension of 10 kN, which is applied by tightening the nut with torque. We want to know the maximum stress at the threads under such a pretension.

Pretension is a built-in environment condition in Workbench 3D simulations, in which a pretension can be applied to a body or cylindrical surface. It is, however, not applicable for 2D simulations.

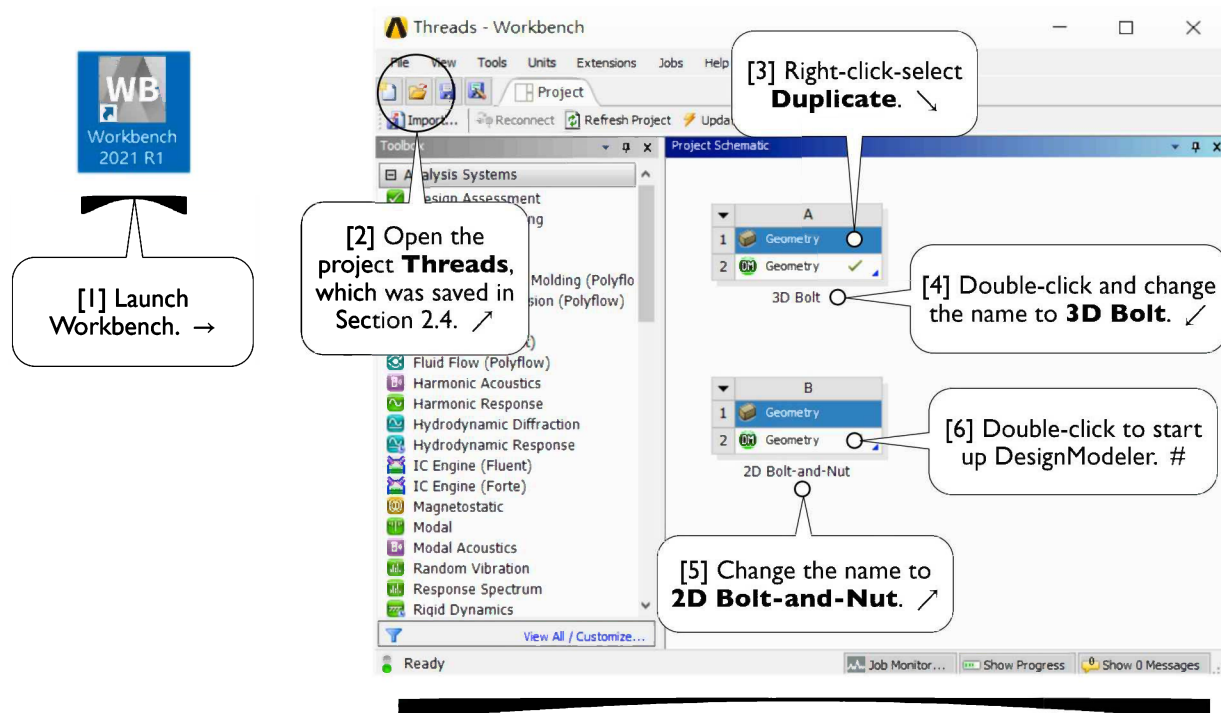
In this section, we will make some simplifications for a 2D simulation. Assuming a mirror-symmetry between the upper half and the lower half (which is not exactly true), we model only the upper part of the assembly [6-8]. To reduce the problem size further and alleviate contact nonlinearity, the plate is not included and its contact with the nut is modeled with a frictionless support [9].

The pretension is modeled using a uniform force applied on the lower face of the bolt [8]. The results will somewhat deviate from the reality, to be discussed at the end of this section, but the deviation has little effect on the stresses of the threads.

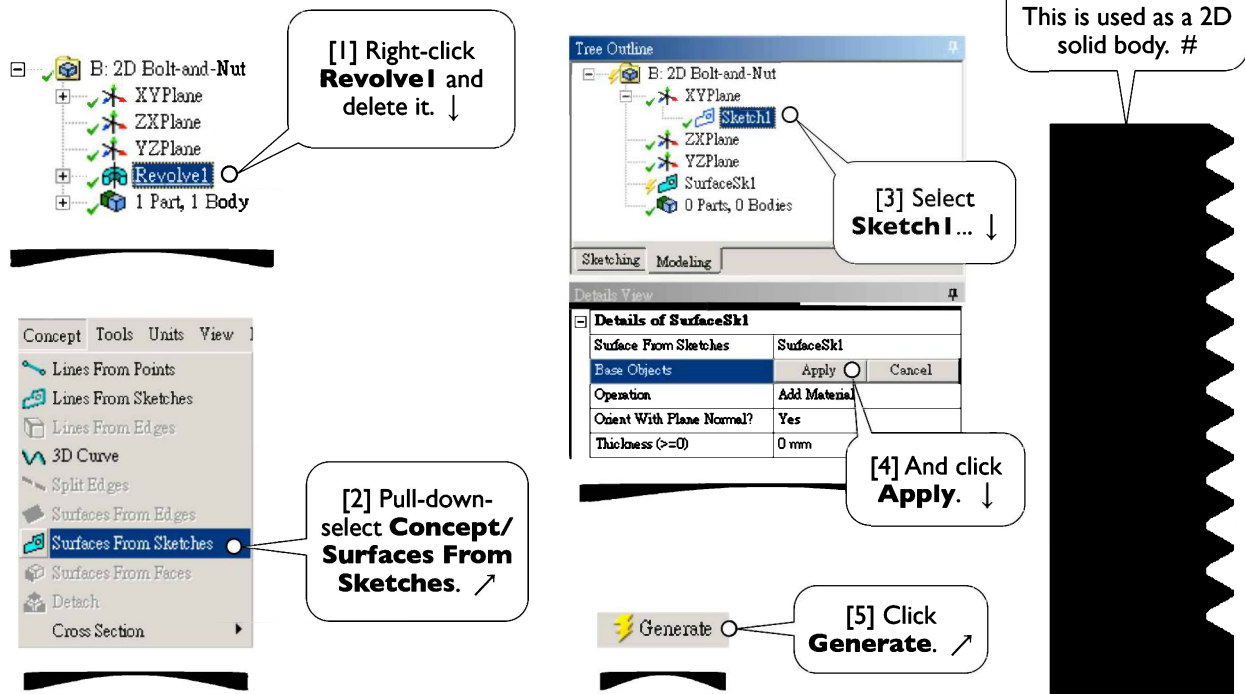
Assume that the coefficient of friction between the bolt and the nut is 0.3. ↗



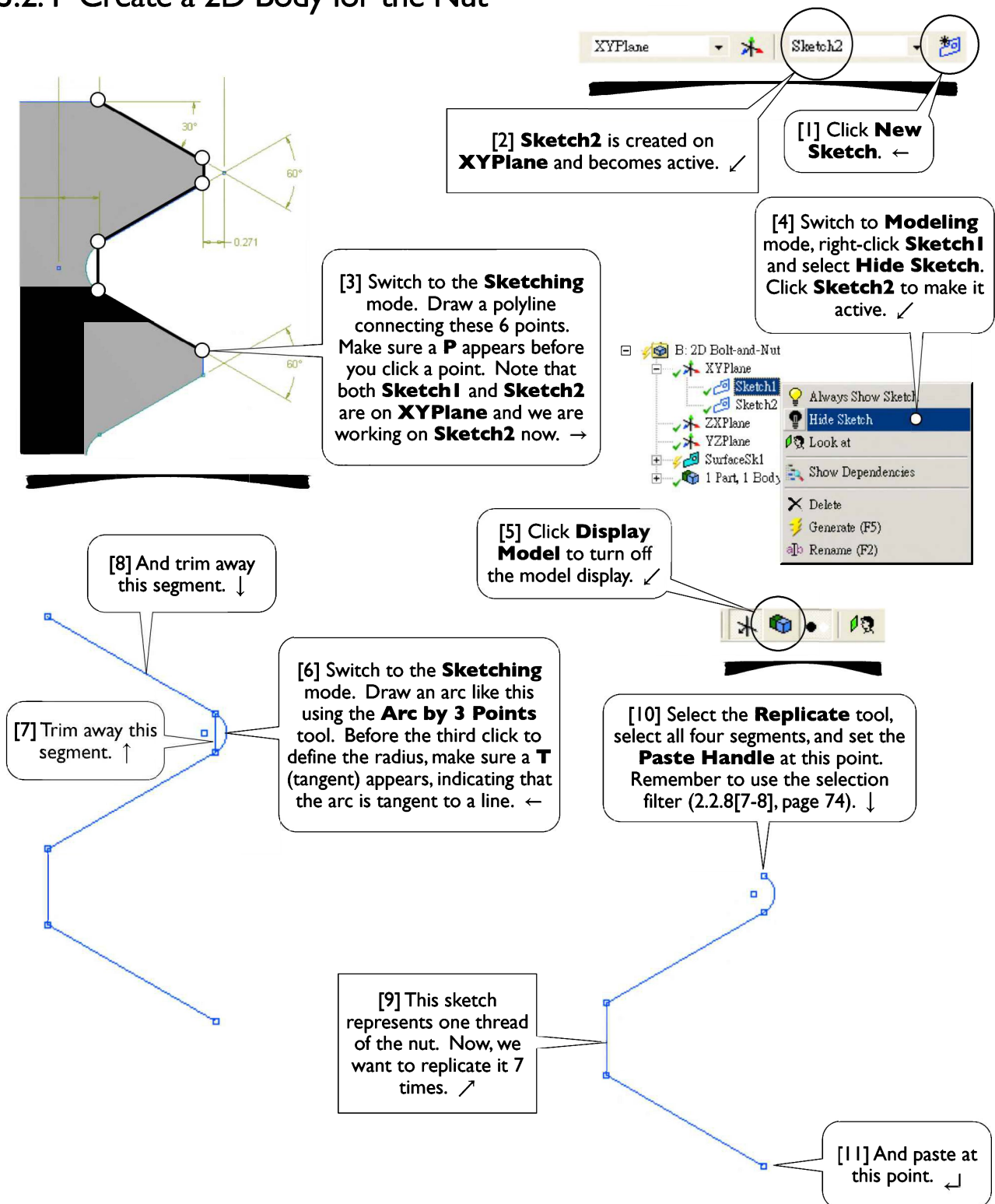
3.2.2 Open the Project **Threads**

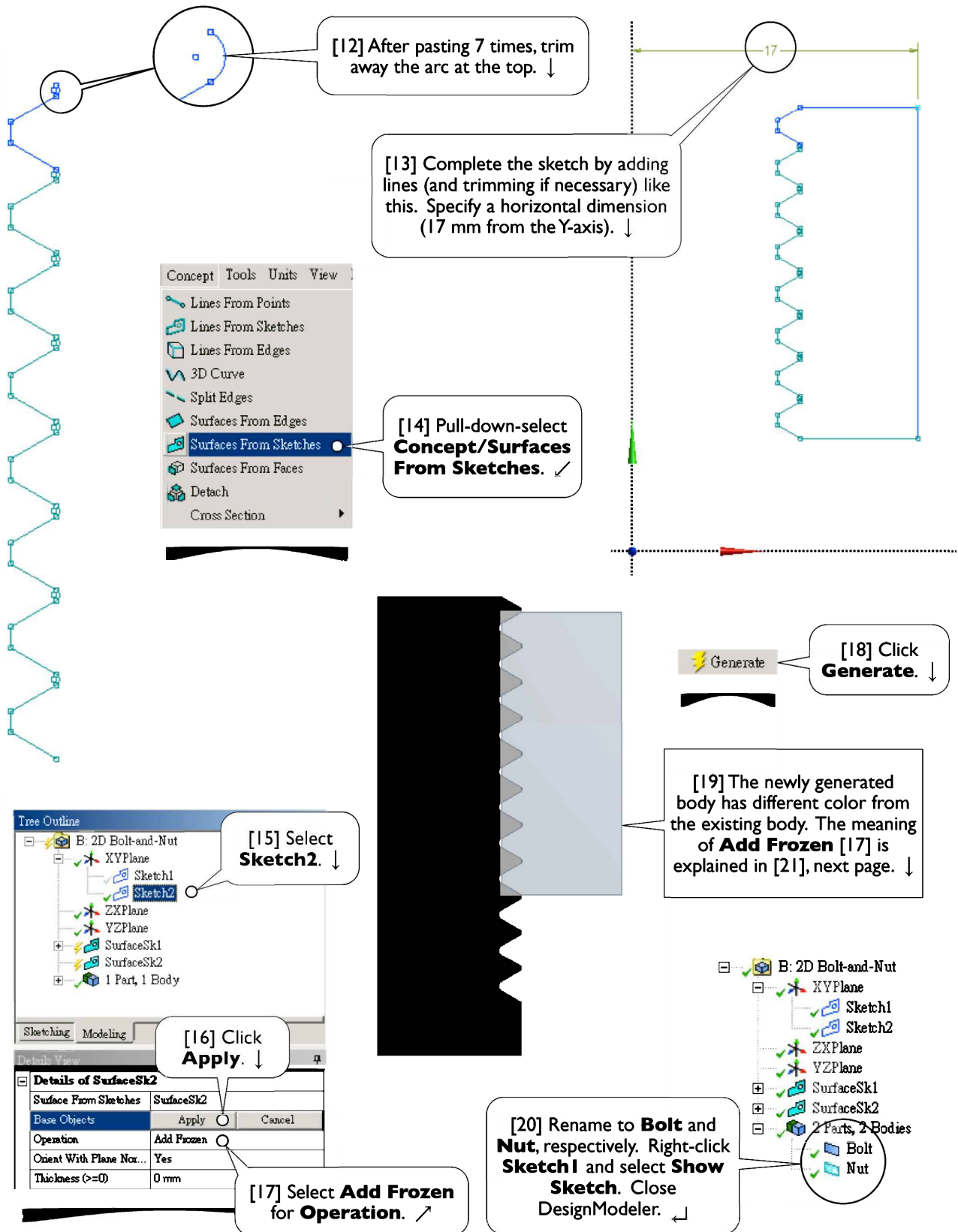


3.2.3 Delete the 3D Body and Create a 2D Body



3.2.4 Create a 2D Body for the Nut





Add Material versus Add Frozen

[21] With **Add Material** operation, the created material adds to the existing body and they become an integral part. On the other hand, if you choose **Add Frozen** ([17], last page), the created material does not add to the existing one; it becomes another part. This is what we intend: the bolt and nut are separate parts; they are not bonded to each other. In Workbench, each part is meshed independently. #

3.2.5 Create an Analysis System

[1] Double-click **Static Structural**. ↑

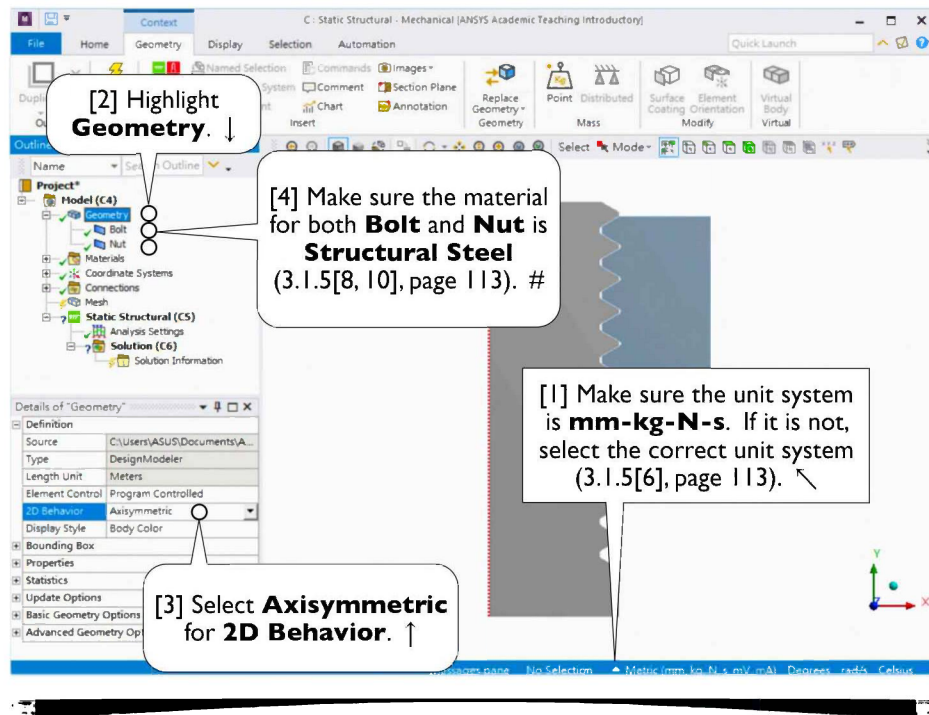
[2] Drag **Geometry...** ↘

[3] And drop to the **Geometry** of **Static Structural**. A link is created. The two systems now share the same **Geometry**. Note that you can edit the upstream **Geometry** cell but not the downstream **Geometry** cell. ↓

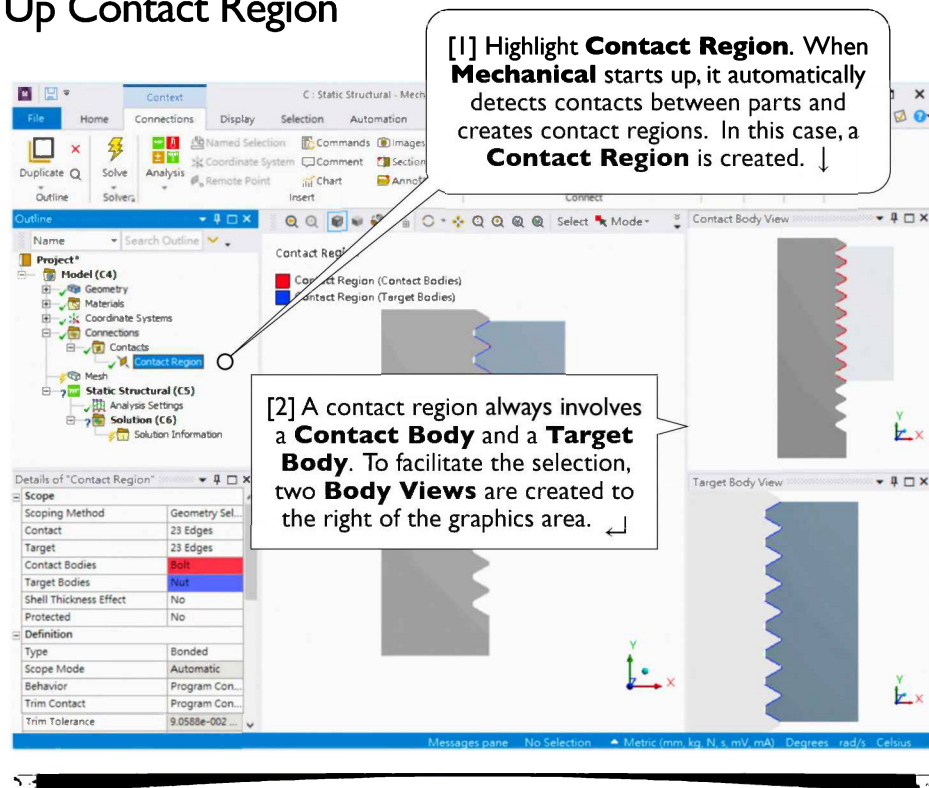
[4] Right-click **Geometry** and select **Properties** from the context menu. Set **Analysis Type** to **2D** (3.1.4[5-6], page 112). ✓

[5] Double-click to start up **Mechanical**. #

3.2.6 Set Up Geometry in Mechanical



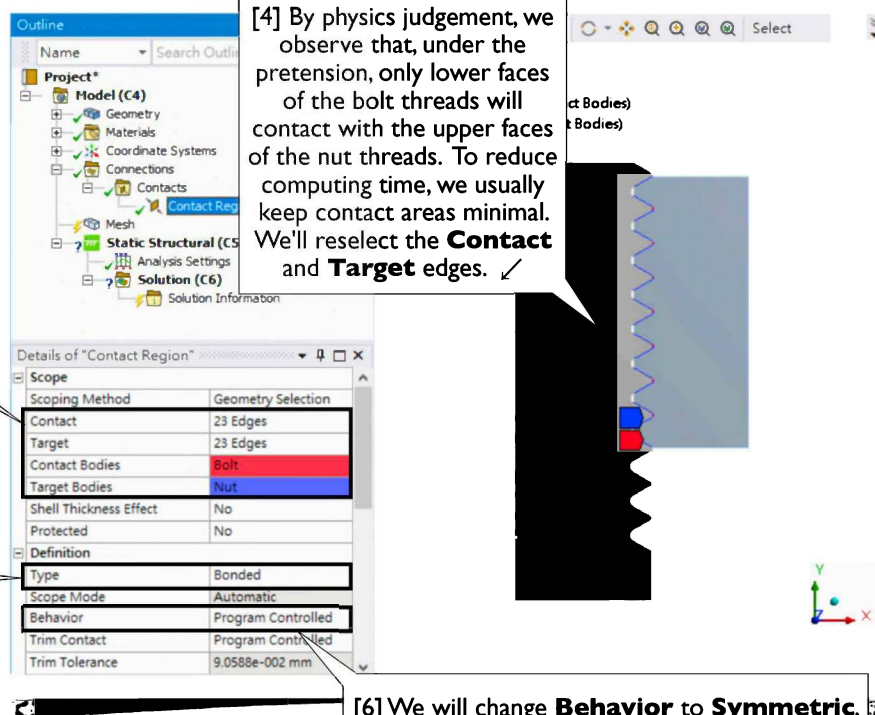
3.2.7 Set Up Contact Region



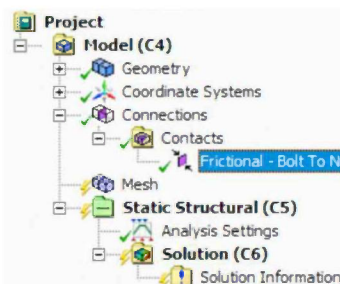
[3] Details of "Contact Region" shows that 23 edges of the bolt are detected and treated as **Contact** and 23 edges of the nut are detected and treated as **Target**. We'll remove some edges that won't contact, to save computing time. →

[4] By physics judgement, we observe that, under the pretension, only lower faces of the bolt threads will contact with the upper faces of the nut threads. To reduce computing time, we usually keep contact areas minimal. We'll reselect the **Contact** and **Target** edges. ✓

[5] We also want to change **Type** from **Bonded** to **Frictional** (13.1.8, page 476). →



[6] We will change **Behavior** to **Symmetric**, meaning that the contact and target are treated symmetrically, so we don't need to decide which one is contact and which one is target (13.1.9, page 477). ✓



[8] The name automatically changes to reflect the contact type. ↓

[7] Select **Frictional** for **Type**. ↑

[9] Type 0.3 for **Friction Coefficient**. ↓

[10] Select **Symmetric** for **Behavior**. ↓

[11] Select **Augmented Lagrange** for **Formulation**. It is recommended for frictional or frictionless contacts (13.1.10, pages 477-478). ↙

[12] From the **Graphics Toolbar**, click the **Edge** filter. →

[13] In the **Contact Body View** (remember that the bolt is the **Contact Body**), control-select these lower edges of bolt-thread, 8 edges in total. You may enlarge the view to facilitate the selection. ←

[14] Click to bring up **Apply/Cancel** buttons and then click **Apply**. This redefines the 8 edges as **Contact**. ↘

Details of "Frictional - Bolt To Nut"	
Scope	
Scoping Method	Geometry Selection
Contact	8 Edges
Target	23 Edges
Contact Bodies	Bolt
Target Bodies	Nut
Shell Thickness Effect	No

[15] In the **Target Body View** (remember that the nut is the **Target Body**), control-select all the upper edges of the nut-thread, 8 edges in total. ←

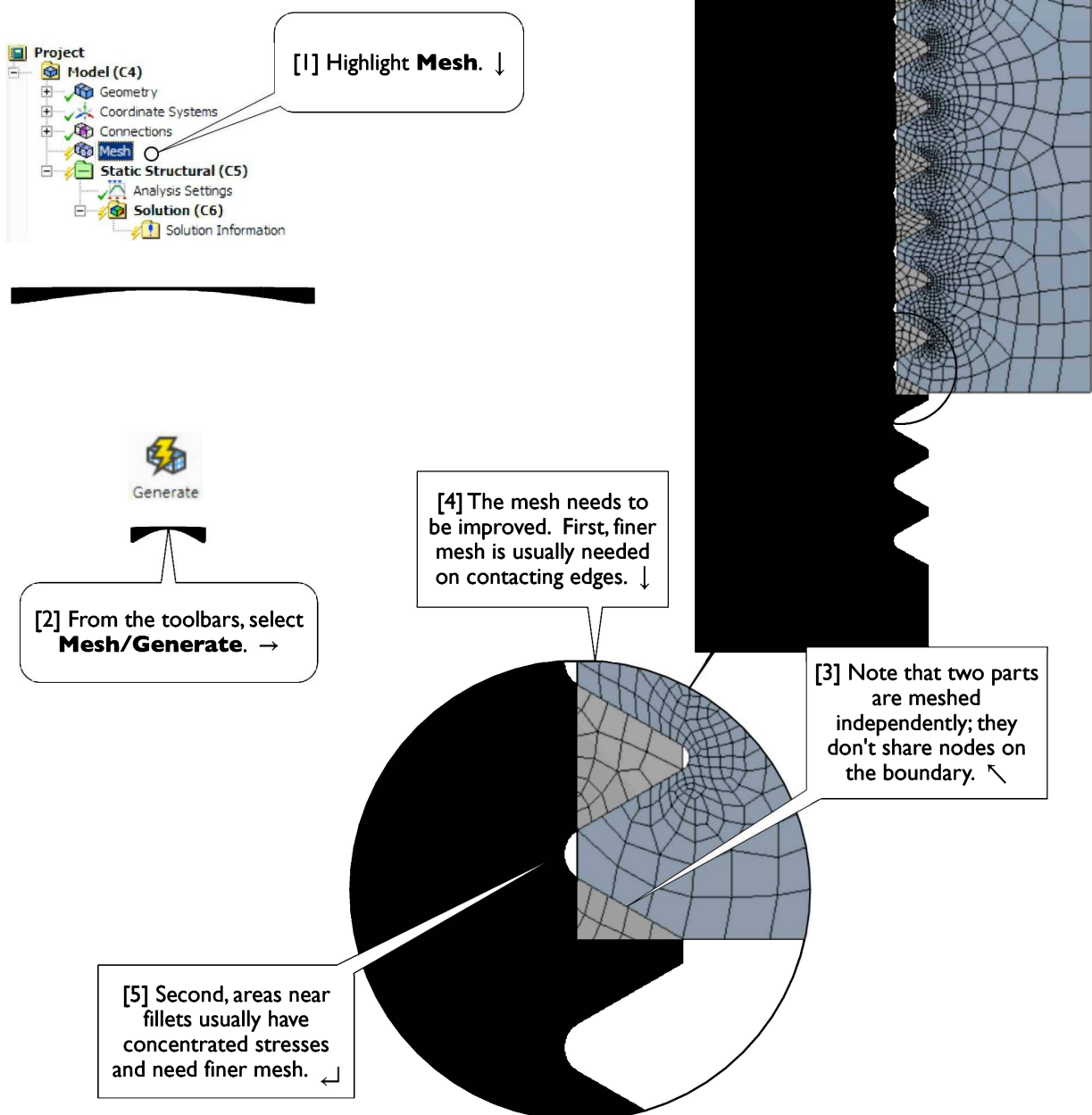
[16] Click to bring up **Apply/Cancel** buttons and then click **Apply**. This redefines the 8 edges as **Target**. ↙

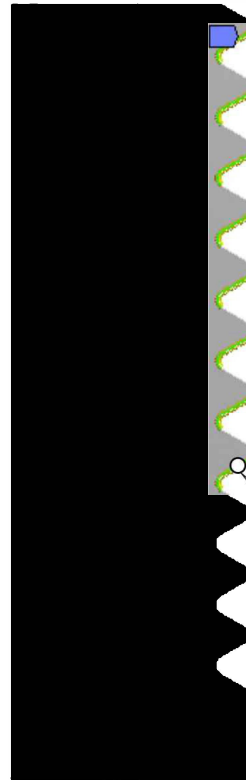
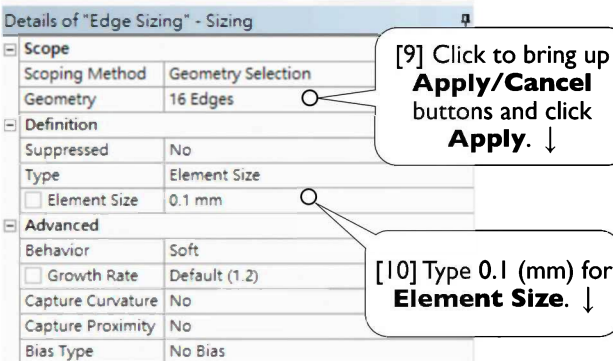
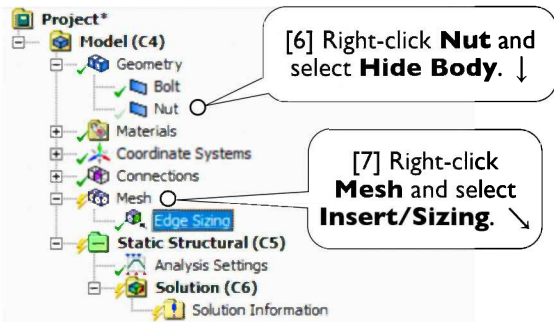
Details of "Frictional - Bolt To Nut"	
Scope	
Scoping Method	Geometry Selection
Contact	8 Edges
Target	8 Edges
Contact Bodies	Bolt
Target Bodies	Nut
Shell Thickness Effect	No

Contact Elements

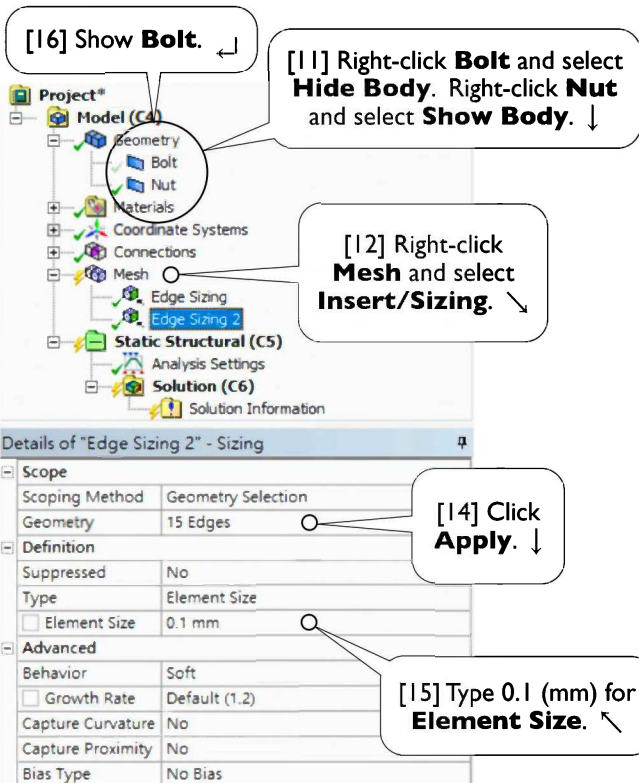
[17] During mesh generation (3.2.8, this page), Workbench creates contact elements between the contact edges and the target edges. Contact elements are used to prevent a *contact body* from penetrating into a target body. Therefore, you should set up contact regions wherever contacts may occur. As long as the behavior is **Symmetric** ([10], page 131), you may choose any one as contact body and the other as target body. PART B of Chapter 13 discusses contacts in detail (pages 476-479). #

3.2.8 Generate Mesh

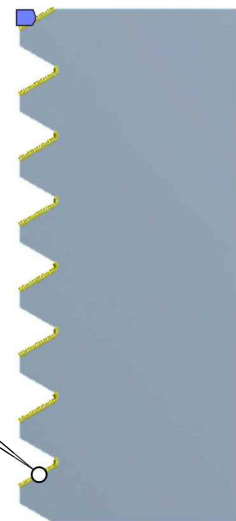




[8] Control-select the lower edges and the fillets of the upper 8 threads, 16 edges in total. You may need to use **Edge filter**. ←



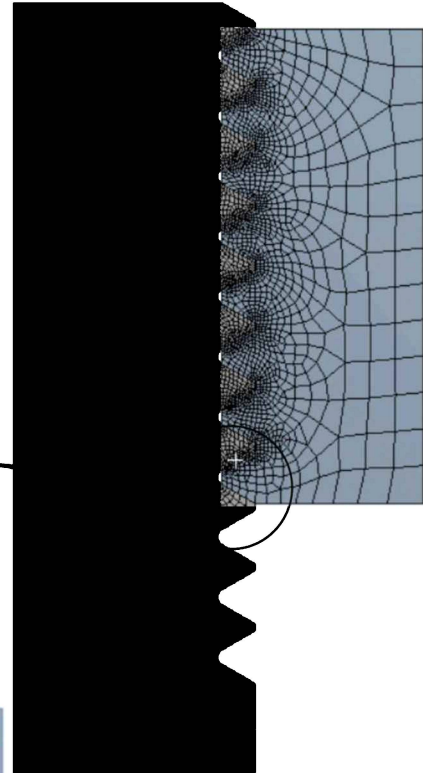
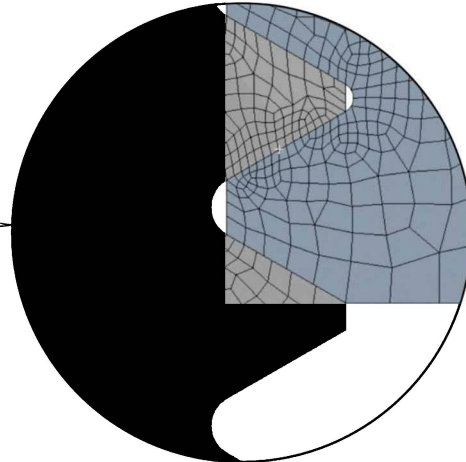
[13] Control-select all the upper edges and all the fillets, 15 edges in total. ←



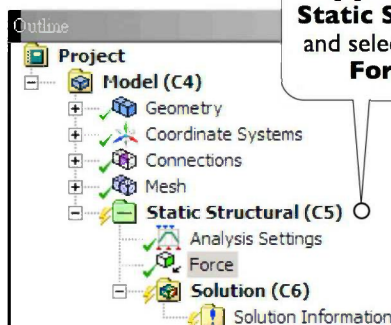


[17] With **Mesh** selected, click **Mesh/Generate**. ✓

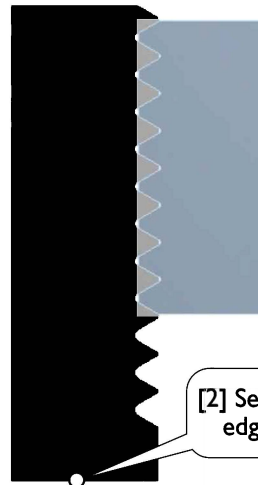
[18] Remember that, in general, the solution is more accurate when the mesh is finer. #



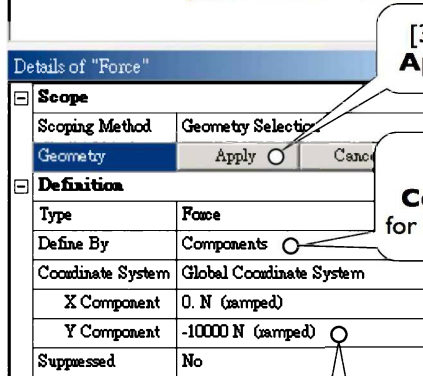
3.2.9 Specify Loads



[1] Right-click **Static Structural** and select **Insert/Force**. ↘



[2] Select this edge. ←



[3] Click **Apply**. ↓

[4] Select **Components** for **Define By**. ↓

[5] Type -10000 (N) for **Y Component**. →

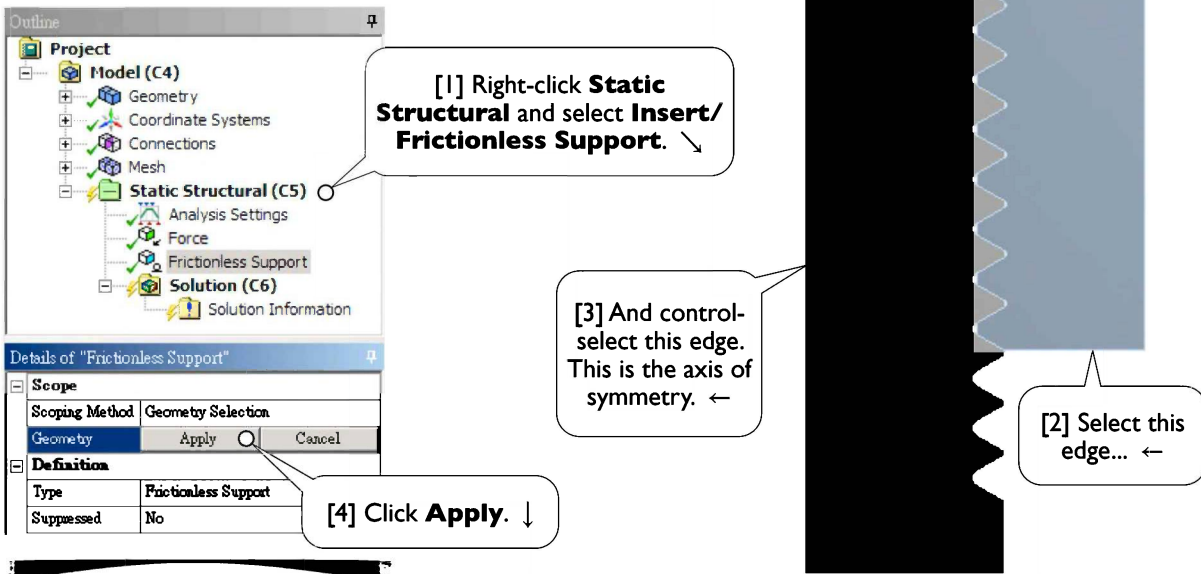
How is the force distributed?

[6] Specifying a force of 10,000 N on the bottom edge of the axisymmetric 2D model means that the force is distributed evenly on the face created by revolving the edge 360°. In this case, it has exactly the same effect if you specify a (negative) **pressure** of

$$\frac{10000}{(\pi \times 20^2 / 4)} = 31.83 \text{ MPa}$$

#

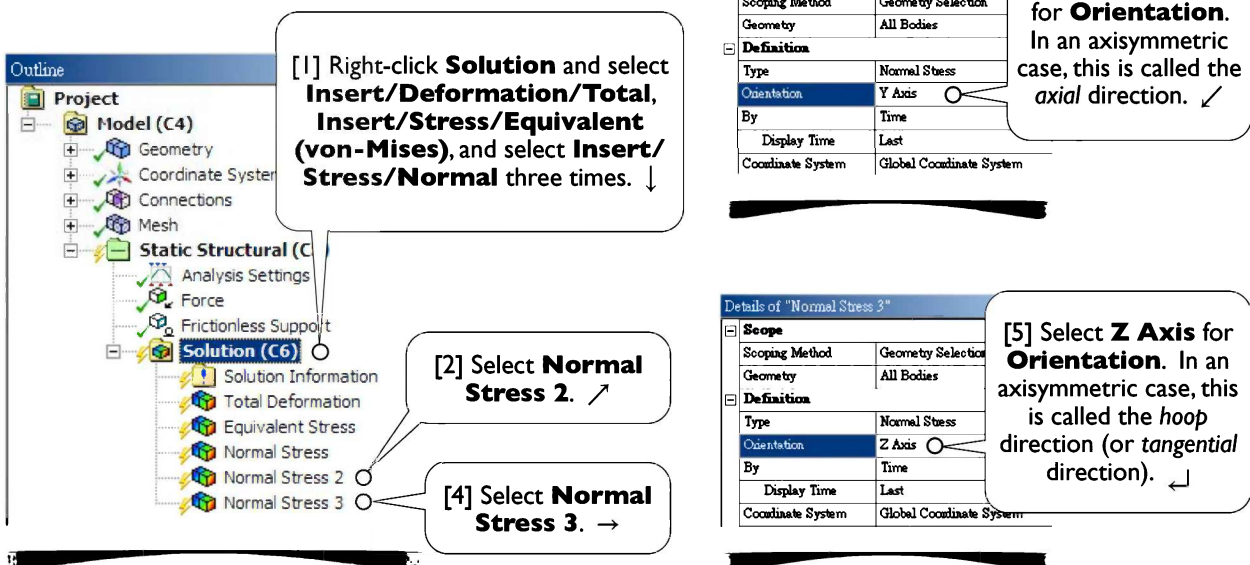
3.2.10 Specify Supports

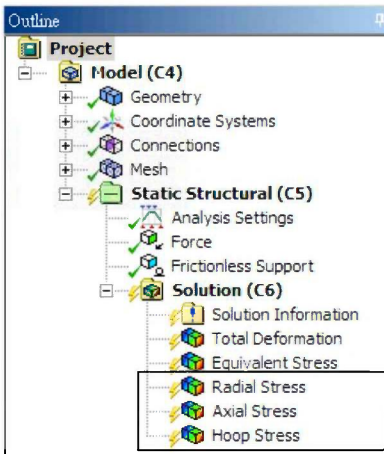


Boundary Conditions for the Axis of Symmetry

[5] Since any point on an axis of symmetry (here, Y-axis) does not move in the radial-direction (X-direction), you must specify a zero X-displacement condition or, equivalently, a frictionless support on the axis of symmetry [3]. Some FEA software can automatically take care of this boundary condition; however, as a good practice, always explicitly specify this boundary condition. If you leave it as a free boundary, the axis may become a small cylindrical "hole" after deformation. #

3.2.11 Solve the Model

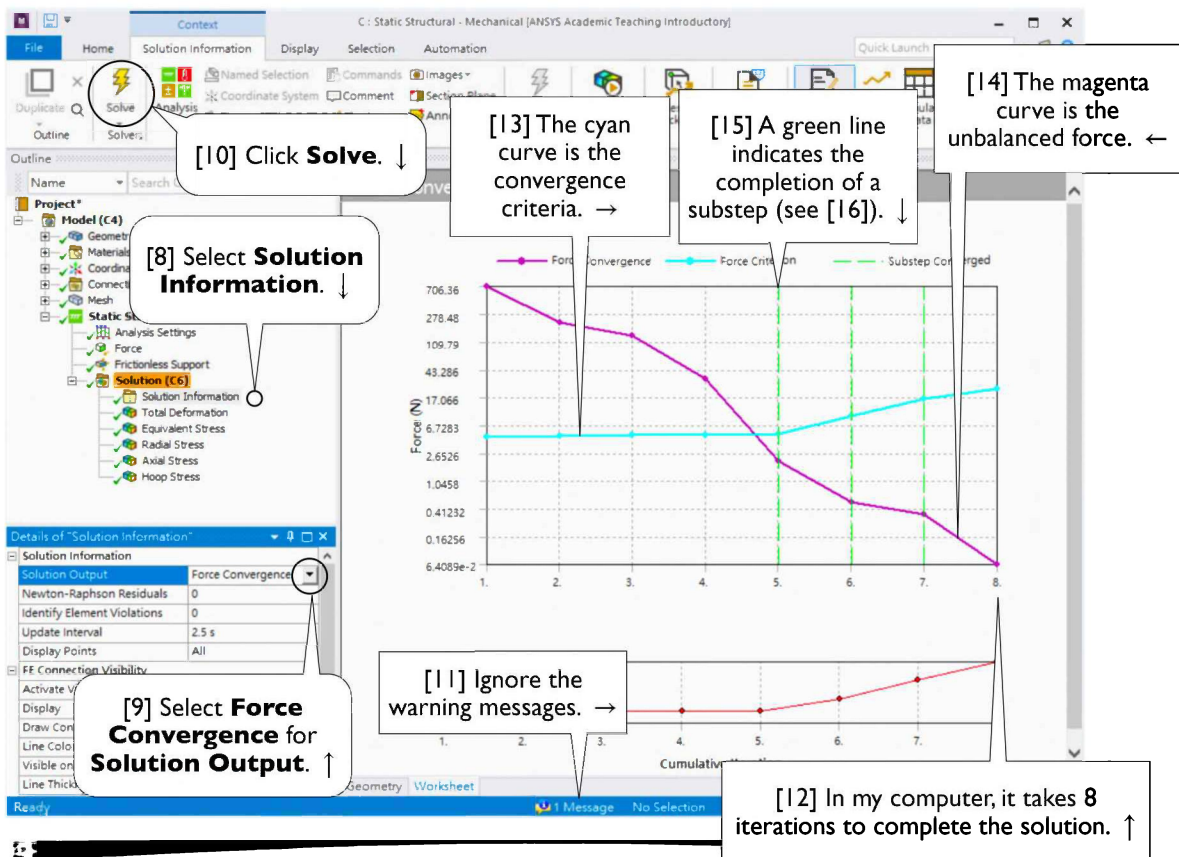




[6] Rename (by right-click-selecting **Rename**) these 3 result objects like this. ↑

Nonlinear Simulations

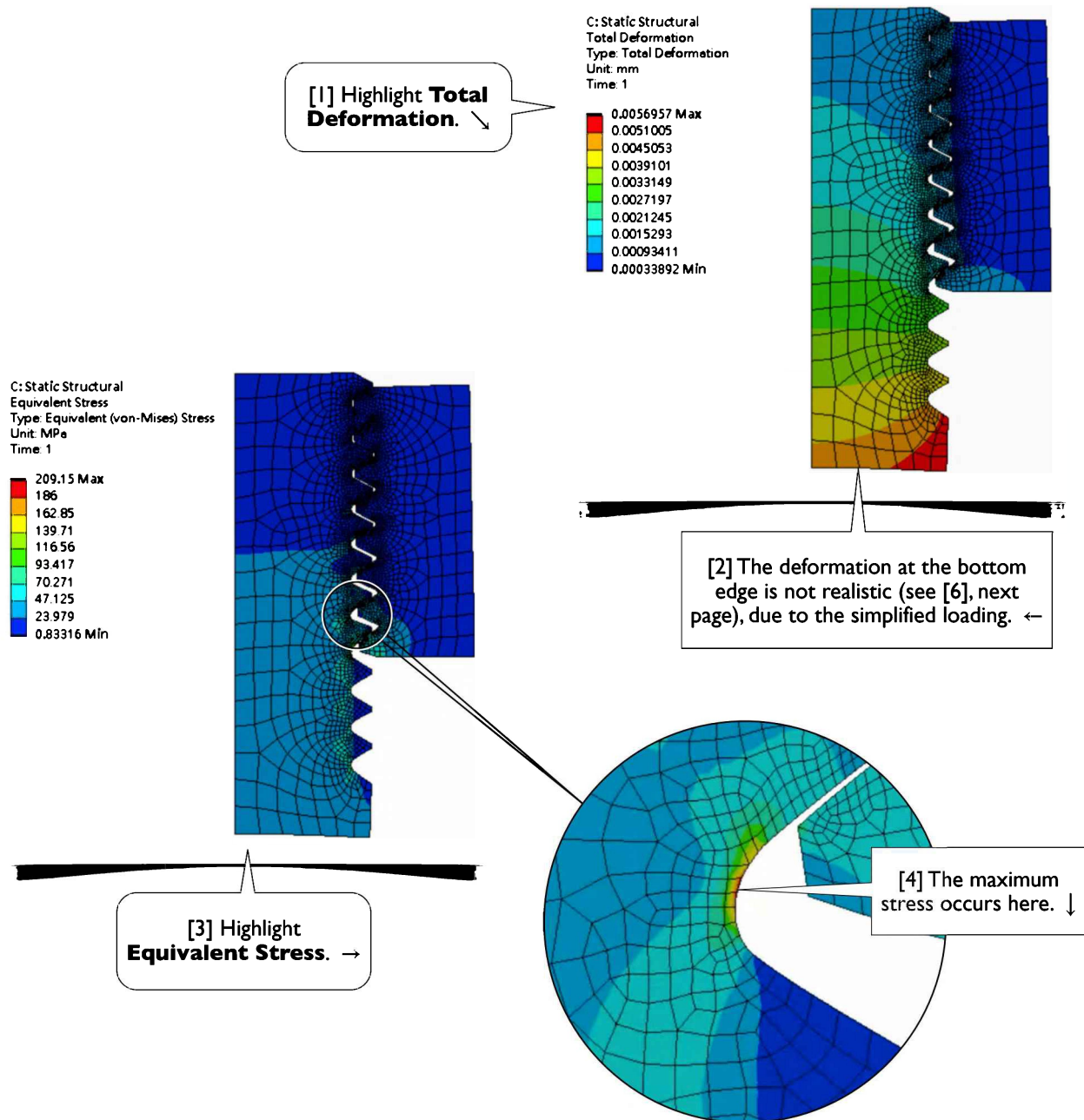
[7] Due to the contact status, as the loads increase, the structure's stiffness also changes. The simulation is no longer linear. Workbench will solve the model using nonlinear solution methods. We will discuss nonlinear solution methods in Chapters 13-14. For now, to gain a feeling of nonlinear solution, let's watch how the solution proceeds [8-16]. ✓



Nonlinear Solution Method: Force Convergence

[16] In this case, Workbench divides the load (10,000 N) into 4 substeps [15] (i.e., increasing 2,500 N each substep). The cyan curve is the convergence criteria [13] and the magenta curve is the "unbalanced" force [14]. A substep converges when the unbalanced force is less than the criterion [13-15]. PART A of Chapter 13 (pages 469-475) will discuss the details. #

3.2.12 View the Results



2D Models Must Be in XY Plane^[Ref 1]

[5] ANSYS requires that, for a 2D problem (i.e., plane-stress, plane-strain, or axisymmetric problems), the 2D geometric model **MUST** lie in the global XY plane. Besides, for an axisymmetric problem, the global Y-axis **MUST** be the axis of symmetry and the model **MUST** be placed on the +X side.

A surface body created in DesignModeler is called a "2D solid body" if it is used in **Mechanical** for a 2D simulation, in which 2D solid elements (1.3.3[6-8], page 38) are used. ↵

Modeling Considerations

[6] The bottom edge ([2], last page) is actually a plane of symmetry; it must remain horizontal and must have no vertical displacements. This plane of symmetry might have been modeled as a frictionless support; however, a frictionless support cannot have an out-of-plane force acting on it. With this dilemma, we choose to apply a force, and expect an unrealistic deformation at the plane of symmetry [2]. Since we are only concerned about the stress at the threads, and the region of influence of this faulty boundary condition seems not so large as to reach the areas that concern us [4], we decide to accept this arrangement. ↓

Remark

[7] The quantities of bolt-and-nut used in daily industrial applications are huge. Their behavior should be carefully investigated. In our preliminary study, it shows that the stresses are distributed so unevenly that most of the stresses are taken by a few lower contacting threads. To improve the efficiency of the bolt-and-nut, one way is to allot some of the stresses to the upper contacting threads. In his books^[Refs 2, 3], Zahavi has provided several alternatives to reduce the maximum stress. This case is adapted based on an example in his books. ↓

Wrap Up

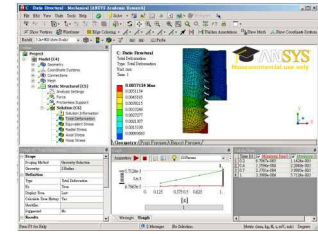
[8] View other results. Close **Mechanical**, save the project, and exit Workbench. #

References

1. All Help>Mechanical APDL>Theory Reference>13.182. PLANE182.
2. Zahavi, E. and Barlam, E., *Nonlinear Problems in Machine Design*, CRC Press LLC, 2000; Chapter 10. Threaded Fasteners.
3. Zahavi, E., *The Finite Element Method in Machine Design*, Prentice-Hall, 1992; Chapter 7. Threaded Fasteners.

Section 3.3

More Details



3.3.1 Plane-Stress Problems

Plane-Stress Condition

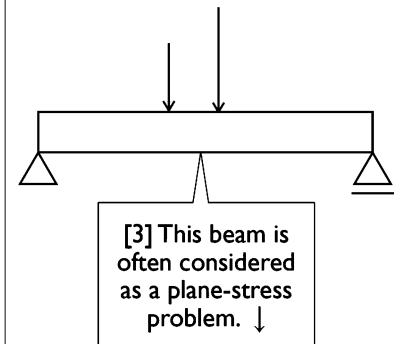
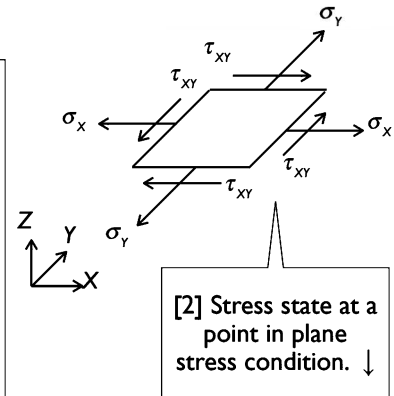
[1] Consider a plate of ZERO thickness on XY plane subject to in-plane forces. The stress state at any point can be depicted in [2]. Note that there are no stresses in Z-face; i.e.,

$$\sigma_z = 0, \quad \tau_{zy} = 0, \quad \tau_{zx} = 0 \quad (1)$$

Eq. (1) is called a plane-stress condition. If the plane-stress condition holds everywhere, then it is called a plane-stress problem.

In the real world, there is no such thing as zero thickness. The triangular plate simulated in Section 3.1 is close to but not exactly a plane-stress problem; the triangular plate has finite thickness of 10 mm. However, since its stresses in Z-direction are negligible, we usually assume that the plane-stress condition holds for such a finite thickness plate.

In practice, a problem may assume the plane-stress condition if its thickness direction (Z-direction) is not restrained and thus free to expand or contract. As an example, a simply supported beam as shown in [3] is often solved by assuming the plane stress condition, even though its out-of-plane thickness is not zero.



Hooke's Law for Plane-Stress Problems

Substituting the plane-stress condition, Eq. (1), into Eq. 1.2.8(1) (page 31), Hooke's law becomes

$$\begin{aligned} \varepsilon_x &= \frac{\sigma_x}{E} - \nu \frac{\sigma_y}{E} \\ \varepsilon_y &= \frac{\sigma_y}{E} - \nu \frac{\sigma_x}{E} \\ \varepsilon_z &= -\nu \frac{\sigma_x}{E} - \nu \frac{\sigma_y}{E} \\ \gamma_{xy} &= \frac{\tau_{xy}}{G}, \quad \gamma_{yz} = 0, \quad \gamma_{zx} = 0 \end{aligned} \quad (2)$$

Substitution of Eq. (1) into other governing equations (e.g., Eq. 1.2.6(2), page 30) will conclude that all quantities are independent of Z. That is, the particles with the same X and Y coordinates share the same behaviors regardless of their Z coordinate. Thus, we can eliminate the Z coordinate and reduce the problem to a two-dimensional problem, on XY space.

Note that, in Eq. (2), ε_z is not zero; it can be calculated from σ_x and σ_y . The nonzero ε_z is easy to understand: since Z-direction is free to expand or contract, there must be strains in the Z-direction due to poisson's effect. ↗

2D models must be in XY plane^[Ref 12]

[4] ANSYS requires that, for a 2D problem (i.e., plane-stress, plane-strain, or axisymmetric problems), the 2D geometric model MUST lie in a global XY plane. For an axisymmetric problem, the global Y-axis is always the axis of symmetry and the model MUST be placed on the +X side. (Also see 3.2.12[5], page 138.) #

3.3.2 Plane-Strain Problems

Plane-Strain Condition

[1] Consider a structure of INFINITE LENGTH in Z-direction. The Z-direction is restrained such that no particles can move in Z-direction. Further, all cross-sections perpendicular to the Z-direction have the same geometry, supports, and loads [2]. In such a case, the strain state at any point can be depicted in [3]. Note that there are no strains in the Z-face; i.e., $\varepsilon_z = 0$ (otherwise the particle on the Z-face would move in Z-direction) and $\gamma_{zx} = \gamma_{zy} = 0$ (otherwise the cube would twist in ZX and ZY planes respectively, and that implies the particles would move in Z-direction),

$$\varepsilon_z = 0, \quad \gamma_{zx} = 0, \quad \gamma_{zy} = 0 \quad (1)$$

Eq. (1) is called a *plane-strain condition*. If the plane-strain condition holds everywhere, then it is called a plane-strain problem.

In the real world, there is no such thing as infinite length. In practice, a problem may assume the plane-strain condition if its Z-direction is restrained from expansion or contraction and all cross-sections perpendicular to the Z-direction have the same geometry, supports, and loads. As an example, a pressurized pipe buried under the earth is often considered as a plane-strain problem. Section 14.3 and the exercise problems in 3.6.2 (page 171) provide two examples for plane-strain problems.

Hooke's Law for Plane-Strain Problems

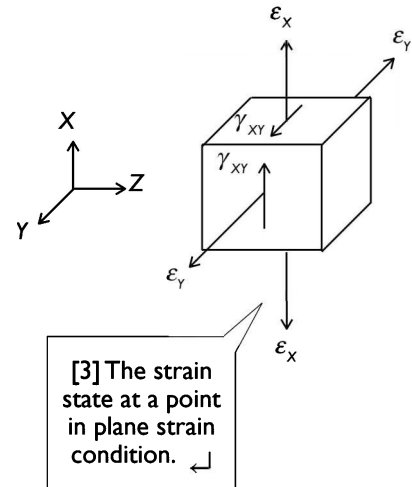
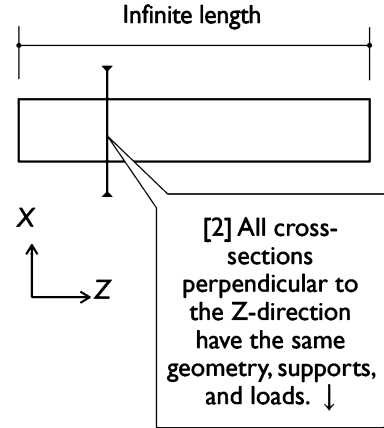
Eq. 1.2.8(1) (page 31), Hooke's law, can be inverted and rewritten as

$$\begin{aligned} \sigma_x &= \frac{E}{(1+\nu)(1-2\nu)} [(1-\nu)\varepsilon_x + \nu\varepsilon_y + \nu\varepsilon_z] \\ \sigma_y &= \frac{E}{(1+\nu)(1-2\nu)} [(1-\nu)\varepsilon_y + \nu\varepsilon_z + \nu\varepsilon_x] \\ \sigma_z &= \frac{E}{(1+\nu)(1-2\nu)} [(1-\nu)\varepsilon_z + \nu\varepsilon_x + \nu\varepsilon_y] \\ \tau_{xy} &= G\gamma_{xy}, \quad \tau_{yz} = G\gamma_{yz}, \quad \tau_{zx} = G\gamma_{zx} \end{aligned} \quad (2)$$

The proof of Eq. (2) is in 3.3.14 (page 150).

Substitute the plane-strain condition Eq. (1) into Eq. (2), and Hooke's law becomes

$$\begin{aligned} \sigma_x &= \frac{E}{(1+\nu)(1-2\nu)} [(1-\nu)\varepsilon_x + \nu\varepsilon_y] \\ \sigma_y &= \frac{E}{(1+\nu)(1-2\nu)} [(1-\nu)\varepsilon_y + \nu\varepsilon_x] \\ \sigma_z &= \frac{E}{(1+\nu)(1-2\nu)} [\nu\varepsilon_x + \nu\varepsilon_y] \\ \tau_{xy} &= G\gamma_{xy}, \quad \tau_{yz} = 0, \quad \tau_{zx} = 0 \end{aligned} \quad (3) \nearrow$$



[4] Substitution of Eq. (1) into other governing equations (e.g., Eq. 1.2.6(2), page 30) will conclude that all quantities are independent of Z ; i.e., the particles with the same X and Y coordinates share the same behaviors regardless of their Z coordinate. Thus, we can eliminate Z coordinate and reduce the problem to a two-dimensional problem, on XY space.

Note that, in Eq. (3), σ_z is not zero; it can be calculated from ε_x and ε_y . The nonzero σ_z is easy to understand: since Z -direction is restrained from expansion or contraction, the material will develop stress to counteract the restriction. #

3.3.3 Axisymmetric Problems

[1] Consider a structure of which the geometry, supports, and loads are axisymmetric about the Y -axis. In such a case, all quantities are independent of θ coordinate; i.e., the particles with the same R and Y coordinates share the same behaviors regardless of their θ coordinate. Thus, we may eliminate θ coordinate and reduce the problem to a two-dimensional problem, on R - Y space.

The strain state at any point can be depicted in [2]. Note that there are no shear strains in θ -face (otherwise the θR -face and the θY -face would twist and the problem is no longer axisymmetric),

$$\gamma_{\theta R} = 0, \quad \gamma_{\theta Y} = 0 \quad (1)$$

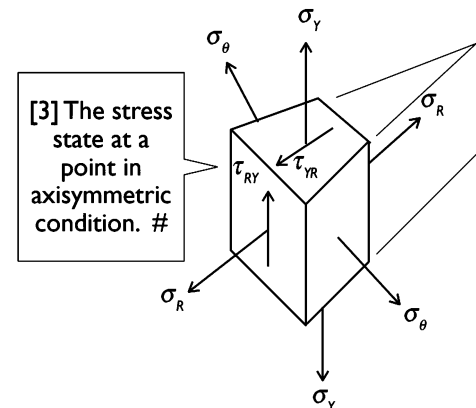
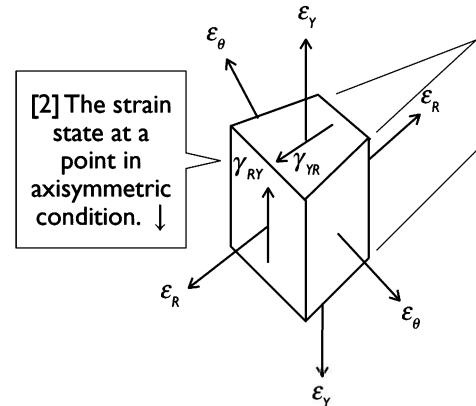
According to Hooke's law, Eq. (1) implies

$$\tau_{\theta R} = 0, \quad \tau_{\theta Y} = 0 \quad (2)$$

Eqs. (1) and (2) can be regarded as the *axisymmetric condition*.

Axisymmetric problems are ubiquitous in engineering applications. Many problems are not strictly axisymmetric but can reasonably assume the axisymmetric condition, such as the bolt-and-nut problem simulated in Section 3.2, in which the threads are spiral and the nut is hexagonal.

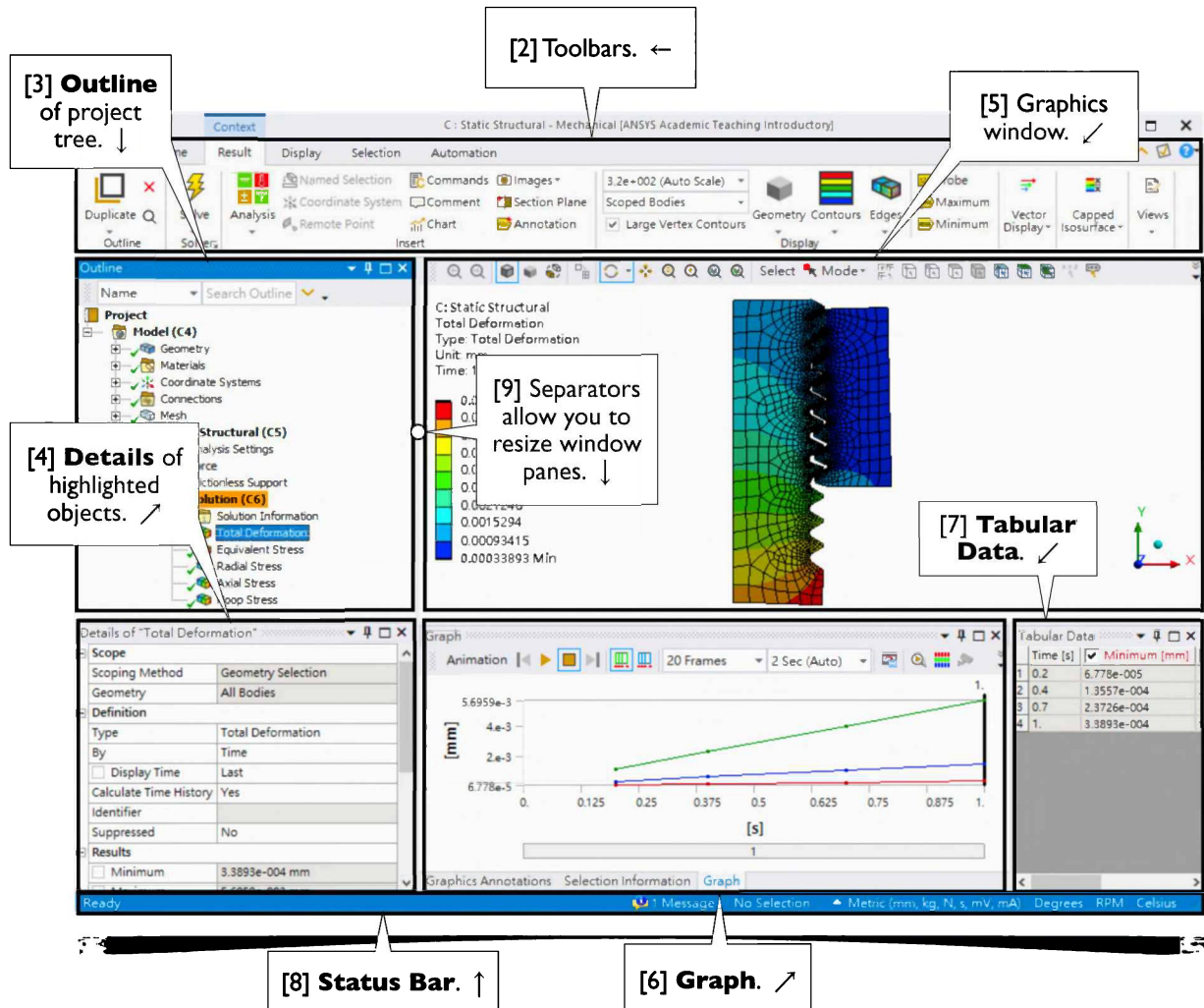
In an axisymmetric problem, σ_r is called the *radial stress*, σ_θ is called the *hoop stress*, and σ_y is called the *axial stress* [3]. ↗



3.3.4 Mechanical GUI

[1] **Mechanical GUI** is composed of several areas ([2-8], next page); many of them are similar to those in **DesignModeler GUI** (2.3.1, page 81). On the top are pull-down menus and toolbars [2]; on the bottom is a status bar [8]. In-between are several "window panes" [3-7]. The separators [9] between window panes may be dragged to resize window panes. You even can move or dock a pane by dragging its title bar. Whenever you mess up the workspace, pull-down-select **View/Windows/Reset Layout** to reset to the default layout.

Outline [3] displays an outline of a *project tree*, which is a structured representation of the project (to be discussed). **Details** [4] shows the detail information of the object highlighted in the project tree or graphics window. The graphics window [5] displays a geometric model. **Graph** [6] typically shows a result-versus-time plot. **Tabular Data** [7] shows the numerical data of the result-versus-time plot. A set of animation tools are available in **Graph**; these tools allow you to play, stop, or save the animation. ↵

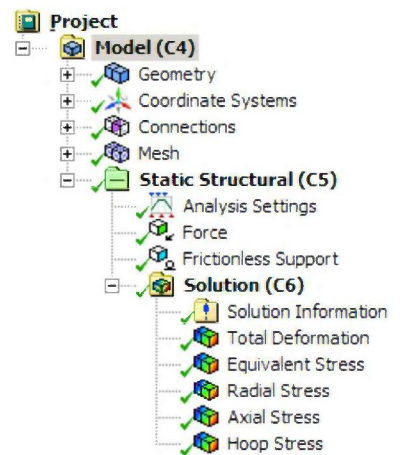


Project Tree^[Refs 1, 11]

[10] A project tree [11] is a structured representation of a project. A project tree may contain one or more **Models**; often, there is only one in a project tree. A **Model** may contain one or more **Environment** branches, along with other objects. Each can be renamed. Default name for an **Environment** branch is the name of the analysis system, for example: **Static Structural**. An **Environment** branch contains **Analysis Settings**, several objects that define the environment conditions, and a **Solution** branch, which contains a **Solution Information** and several results objects.

Right-clicking an object (or multiple objects) and selecting a tool from the context menu, you can operate on the object (or objects), such as delete, rename, duplicate, etc.

Unlike the objects of a model tree in DesignModeler, in which their order is important, the order of the objects in a project tree is not relevant. →



[11] A project tree. #

3.3.5 Unit Systems

[1] In DesignModeler, the only units used are length and angle. In **Mechanical**, units are much more involved; nevertheless, Workbench takes care of the consistency of unit systems, and all you have to do is to select a unit system suitable for your model. Selecting a suitable unit system for your model is crucial in many cases, in which the solution accuracy may deteriorate due to an accumulation of machine errors.

Choosing Unit Systems: Guideline

As a guideline, select a unit system such that the values stored in the computer have about the same order. For example, if you choose SI unit system for a micro-scale simulation model, you would have the lengths of order 10^{-6} and a Young's modulus of order 10^{11} . That may raise precision issues. On the other hand, if you choose a μ MKS unit system, you will have the lengths of order 10^0 and a Young's modulus of order 10^5 . That is much better.

Consistent vs. Inconsistent Unit Systems

In **Workbench GUI** (not **Mechanical GUI**) pull-down-select **Units/Unit Systems...**, and you will see a list of built-in unit systems [2-6]. For internal computation, Workbench always uses a consistent unit system. There are 6 consistent unit systems in the list: SI, CGS, NMM, μ MKS, BIN, and BFT [6]. Highlight a unit system in the list; you will see the details of that unit system.

Other unit systems are *inconsistent* ones. They are, however, often more convenient to use than consistent ones. When you select an inconsistent unit system for use in **Mechanical**, it internally uses a consistent unit system that is closest to the inconsistent one you've chosen.

Like DesignModeler, **Mechanical** allows you to change the unit system any time, using the pull-down menu on **Status Bar** [7]. The internal consistent unit system also changes accordingly. ✓

[2] Built-in unit systems. →

[3] The unit system for the current project. ↓

[4] The default project unit system. ↓

[5] Checked unit systems are NOT available in the pull-down menu. ✓

[6] These, along with the SI, are consistent unit systems. →

[7] **Mechanical** allows you to change the unit system any time, using the pull-down menu on **Status Bar #**

	A	B	C	D
1	Unit System	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
2	SI (kg,m,s,K,A,N,V)	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
3	Metric (kg,m,s,°C,A,N,V)	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
4	Metric (tonne,mm,s,°C,mA,N,mV)	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
5	U.S.Customary (lbm,in,s,°F,A,lbf,V)	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
6	U.S.Engineering (lb,in,s,R,A,lbf,V)	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
7	Metric (g,cm,s,°C,A,dyne,V)	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
8	Metric (kg,mm,s,°C,mA,N,mV)	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
9	Metric (kg,μm,s,°C,mA,μN,V)	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
10	Metric (decatonne,mm,s,°C,mA,N,mV)	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
11	U.S.Customary (lbm,ft,s,°F,A,lbf,V)	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
12	Consistent CGS	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
13	Consistent NMM	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
14	Consistent μ MKS	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
15	Consistent BIN	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
16	Consistent BFT	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
17	DesignModeler Unit System (m, degree)	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
18	DesignModeler Unit System (mm, degree)	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

▲ Metric (mm, kg, N, s, mV, mA)

3.3.6 Environment Conditions

[1] Highlighting an **Environment** branch (e.g., **Static Structural**), you will see a row of environment conditions on the toolbar [2]. These environment conditions may also be accessed through the context menu.

Three groups of environment conditions are frequently used: **Loads** [3], **Supports** [4], and **Inertial** [5]. Environment conditions available in each group depend on the dimensionality (2D or 3D) as well as the type of analysis system (e.g., static or dynamic, structural or thermal).

Here, we will introduce the environment conditions available in 2D static structural simulations. Additional environment conditions will be introduced later, starting from Chapter 5. Many of the environment conditions are self-explanatory while others have many useful features. When going through each environment condition, we will point out its location in the ANSYS documentation system, in which many details can be found. You should consult this official documentation whenever needed.

Inside Workbench

Before jumping to individual environment conditions, let's describe how Workbench processes the environment conditions.

When an environment condition is applied, it will eventually be transferred to the NODES of the finite element model. For example, if you apply a pressure on a surface, the equivalent nodal forces are calculated and applied on nodes. The support conditions are processed in a similar way.

Consider Eq. 1.3.1(1) (page 35) again,

$$[K]\{D\} = \{F\} \quad \text{Copy of Eq. 1.3.1(1)}$$

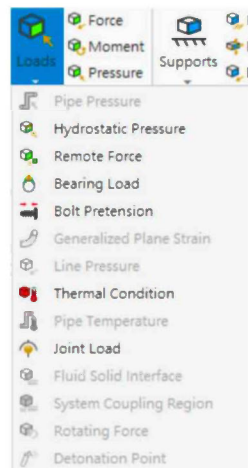
The vector $\{F\}$ is calculated according to the loads and inertia forces, while the vector $\{D\}$ is calculated according to the support conditions. Some of the nodal forces and nodal displacements are known values while the others are unknown. The unknown nodal forces to be solved are called *reaction forces*. For any degree of freedom, if the displacement is known then the corresponding force is unknown, and vice versa.

Magnitude of Environment Conditions^[Ref 2]

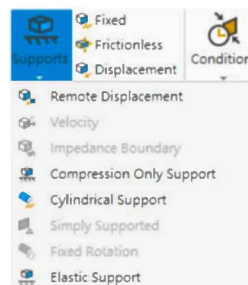
The magnitude of most environment conditions can be specified in three ways: a constant value, a time-dependent tabular form, or a mathematical function with time, X, Y, or Z as the independent variable. ↗



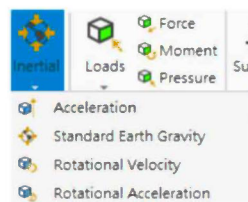
[2] Highlighting an **Environment** branch, you will see a row of available environment conditions on the toolbars. ↓



[3] Loads. ↓



[4] Supports. ↓



[5] Inertial. #

3.3.7 Loads^[Ref 3]

Pressure

[1] Applies to 2D edges or 3D faces. It is possible to define a spatial varying pressure^[Ref 4].

Force

Applies to vertices, edges, or faces. If it applies on edges/faces, the force is evenly distributed on the edges/faces.

Thermal Condition

Applies to bodies. The temperature change ΔT (see Eq. 1.2.8(3), page 32) is the difference between specified temperature and the reference temperature, which is part of the information of the material properties, default to 22°C.

Bearing Load

Applies to 2D circular edges or 3D cylindrical faces. The total force is distributed on the compressive side of the circular edges or cylindrical faces.

Hydrostatic Pressure

Applies to 2D edges or 3D faces. It simulates pressure that occurs due to fluid weight. A free surface location may be specified, default to the surface at $X = 0$.

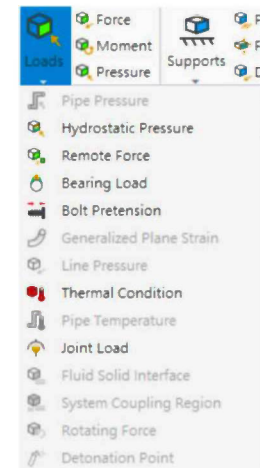
Moment

Applies to 2D edges or 3D faces. A statically equivalent pressure distributed on the edges/faces is calculated and applied on the edges/faces.

Remote Force

Applies at a location anywhere in the space. Workbench calculates the equivalent moment and force and applies them on the body. It may be used as an alternative way of building a rigid part and applying a force on it.

Joint Load



3.3.8 Supports^[Ref 6]

Fixed Support

[1] Applies to vertices, edges, or faces. Prevents nodes from moving in X- Y- and Z-directions. It also prevents nodes from rotations for beam/shell elements.

Displacement

Applies to vertices, edges, or faces. Displacements in X- Y- and Z-directions can be specified. A zero value prevents nodes from moving in that direction. An unspecified value sets that direction free. ↩

Frictionless Support

[2] Applies to 2D edges or 3D faces. Prevents nodes from moving in the normal direction; allows nodes to freely move in the tangential direction.

Compression Only Support

Applies to 2D edges or 3D faces. The associated body is free to depart from the edges or faces, but cannot penetrate them. It in effect sets up a frictionless contact region between the body and a rigid support; it introduces contact nonlinearity into the problem.

Cylindrical Support

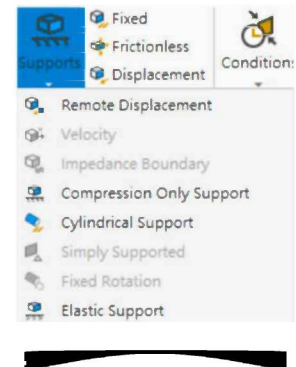
Applies to 2D circular edges or 3D cylindrical faces. Each of the radial, tangential, and axial (3D only) directions can be set free or fixed.

Elastic Support

Applies to 2D edges or 3D faces. A foundation stiffness must be specified to establish the relation between the reaction pressure and the support displacement.

Remote Displacement

Applies at a location anywhere in the space. Workbench calculates the equivalent displacement and rotation and applies them on the body. It may be used as an alternative way of building a rigid part and applying a displacement to it. #



3.3.9 Inertial^[Ref 3]

Standard Earth Gravity

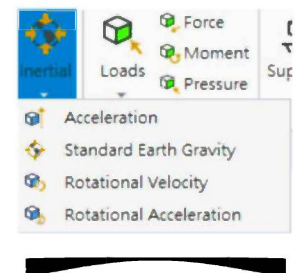
[1] Applies to bodies. You must select a direction along which the gravitational force applies; it defaults to -Z direction.

Acceleration

Applies to bodies. You must specify the magnitude and direction of acceleration. The direction is where the bodies accelerate. An "Inertia force" will apply in the opposite direction.

Rotational Velocity

Applies to bodies. You must specify the magnitude and direction of the angular velocity of the bodies. A distributed "inertia force" will apply in the opposite direction of rotation. #



3.3.10 Results Objects^[Ref 7]



[1] Highlighting **Solution** branch, you will see a row of solution tools available on the toolbars [2]. These tools may also be accessed through the context menu. Most of them are self-explanatory, but some of them need to be explained.

Linearized Stress

Using this tool, you can view stresses along a straight line path. You need to first define a straight line path using **Construction Geometry** under **Model**.


Probe^[Ref 8]

It contains tools to explore the results of a point, or maximum/minimum values of the results of a scoped region, along the loading history. In other words, we are concerned about the results across the time domain instead of space domain.

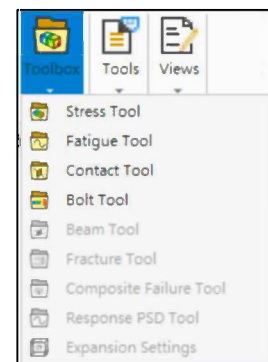
Tools^[Ref 7]

It contains some special solution tools: **Stress Tool**, **Fatigue Tool**, **Contact Tool**, and **Beam Tool** [3].

User Defined Result^[Ref 9]

You may define a result expression using available keywords, which can be listed by highlighting **Solution** in the project tree and clicking the **Worksheet** tab on the right side of the GUI (if you don't see the **Worksheet** tab, click the **Worksheet** toolbar button ). →

[2] Solution toolbar. ↓



[3] Some special solution tools. #

3.3.11 View Results^[Ref 7]

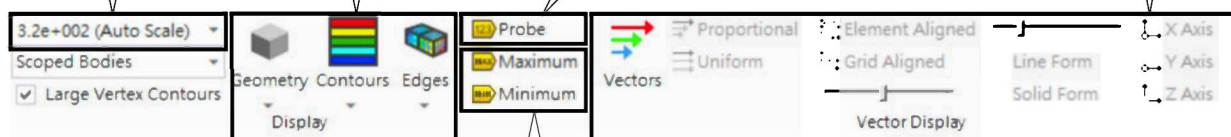
[1] To view results, simply highlight a results object. Tools to control the visual effects of the results are shown on the toolbars [2-6]. ↓

[5] You may select the scale of deformation. →

[6] You can control how the contour displays. #

[3] Click to turn on/off the probe. Results values will display along with your mouse pointer; click to label the value. →

[4] Some results can be displayed in a vector form. ←



[2] Click to turn on/off the label of maximum/minimum. ↑

3.3.12 Insert APDL Commands^[Ref 10]

[1] Clicking **Commands** [2] allows you to insert APDL commands. For those who are familiar with APDL, this may be useful, since Workbench doesn't include all the functionalities provided by APDL. For the newcomers, my suggestion is that you do not worry about APDL for now; Workbench's functionalities are enough for most simulations.

After clicking **Insert Commands** tool, a text editor is opened with several lines of comments telling you WHEN the APDL commands will be executed. For example, an APDL commands object inserted under **Environment** will be executed just before **Solve** command. →

 **Commands**

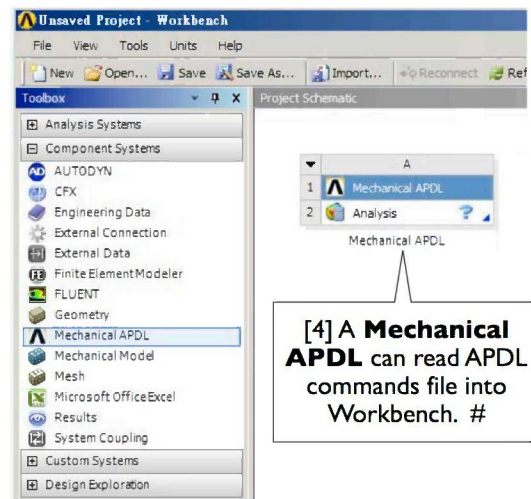
[2] Click **Commands** to insert ANSYS APDL commands. ✓

What is APDL?

[3] In the old days, the users operated ANSYS using a set of text command language, called APDL (ANSYS Parametric Design Language). Compared with modern computer languages, the APDL is not a user-friendly language at all. For many users, use of APDL has been a painful experience.











Later, ANSYS started to provide a graphical user interface (GUI). The users operate ANSYS through pull-down menus, dialogs, etc. Basically, each APDL command has a corresponding operating path in the GUI. Using either APDL or the GUI, the users can use all the functionalities of ANSYS. Again, compared with **Workbench GUI**, the old ANSYS GUI is not efficient at all. Many experts and school teachers prefer APDL to the old GUI.

It is true that some capabilities of APDL are not directly supported in Workbench. It, however, provides two ways that you may access APDL commands: (a) You can insert APDL commands by clicking **Insert Commands** [2]. (b) You can create a **Mechanical APDL** system [4], which allows APDL files to be read into Workbench. →



3.3.13 Status Symbols in Tree Outline^[Ref 11]

[1] Each object of the project tree has a status symbol, explained below:

-  - Checkmark indicates branch is fully defined / OK.
-  - Question mark indicates item has incomplete data (need input).
-  - Lightning bolt indicates solving is required.
-  - Exclamation mark means a problem exists.
-  - "X" means item is suppressed (will not be solved).
-  - Transparent checkmark means body or part is hidden.
-  - Green lightning bolt indicates item is currently being evaluated.
-  - Minus sign means that mapped face meshing failed.
-  - Check mark with a slash indicates a meshed part/body.
-  - Red lightning bolt indicates a failed solution. #

3.3.14 Appendix: Proof of Eq. 3.3.2(2), page 141

[1] The first 3 equations and last 3 equations of Eq. 3.3.2(2) are decoupled; they can be proved independently. Proof of the last 3 equations from the last 3 equations in Eq. 1.2.8(1) (page 31) is trivial. Now, we prove the first 3 equations. The first 3 equations in Eq. 1.2.8(1) can be written in matrix form

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \end{Bmatrix} = \frac{1}{E} \begin{bmatrix} 1 & -\nu & -\nu \\ -\nu & 1 & -\nu \\ -\nu & -\nu & 1 \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \end{Bmatrix} \text{ or } \{\varepsilon\} = [D]\{\sigma\}$$

The first 3 equations in Eq. 3.3.2(2) can also be written in matrix form

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu \\ \nu & 1-\nu & \nu \\ \nu & \nu & 1-\nu \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \end{Bmatrix} \text{ or } \{\sigma\} = [F]\{\varepsilon\}$$

Then

$$[D][F] = \frac{1}{E} \cdot \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & -\nu & -\nu \\ -\nu & 1 & -\nu \\ -\nu & -\nu & 1 \end{bmatrix} \begin{bmatrix} 1-\nu & \nu & \nu \\ \nu & 1-\nu & \nu \\ \nu & \nu & 1-\nu \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

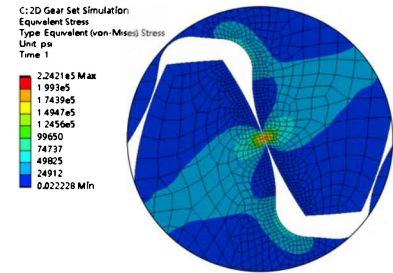
This completes the proof. #

References

1. All Help>Mechanical Application>Mechanical User's Guide>Objects Reference
2. All Help>Mechanical Application>Mechanical User's Guide>Setting Up Boundary Conditions>Defining Boundary Condition Magnitude
3. All Help>Mechanical Application>Mechanical User's Guide>Setting Up Boundary Conditions>Load Type Boundary Conditions
4. All Help>Mechanical Application>Mechanical User's Guide>Setting Up Boundary Conditions>Spatial Varying Loads and Displacements
5. All Help>Mechanical Application>Mechanical User's Guide>Setting Connections>Joints
6. All Help>Mechanical Application>Mechanical User's Guide>Setting Up Boundary Conditions>Support Type Boundary Conditions
7. All Help>Mechanical Application>Mechanical User's Guide>Using Results>Structural Results
8. All Help>Mechanical Application>Mechanical User's Guide>Using Results>Result Output//Probe
9. All Help>Mechanical Application>Mechanical User's Guide>Using Results>User Defined Results
10. All Help>Mechanical Application>Mechanical User's Guide>Commands Objects
11. All Help>Mechanical Application>Mechanical User's Guide>Application Interface>Tree Outline
12. All Help>Mechanical APDL>Theory Reference>13.182. PLANE182

Section 3.4

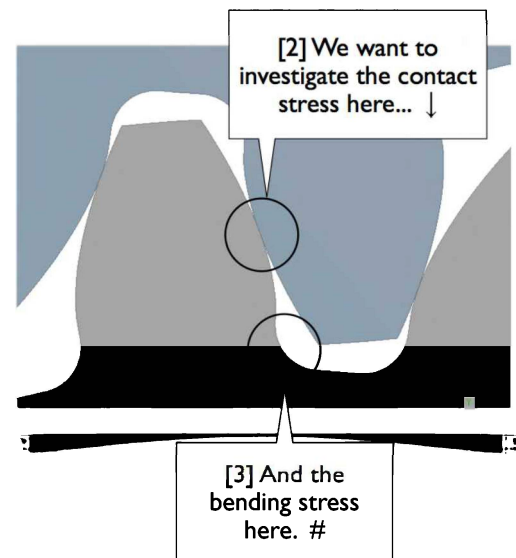
Spur Gears



3.4.1 About the Spur Gears

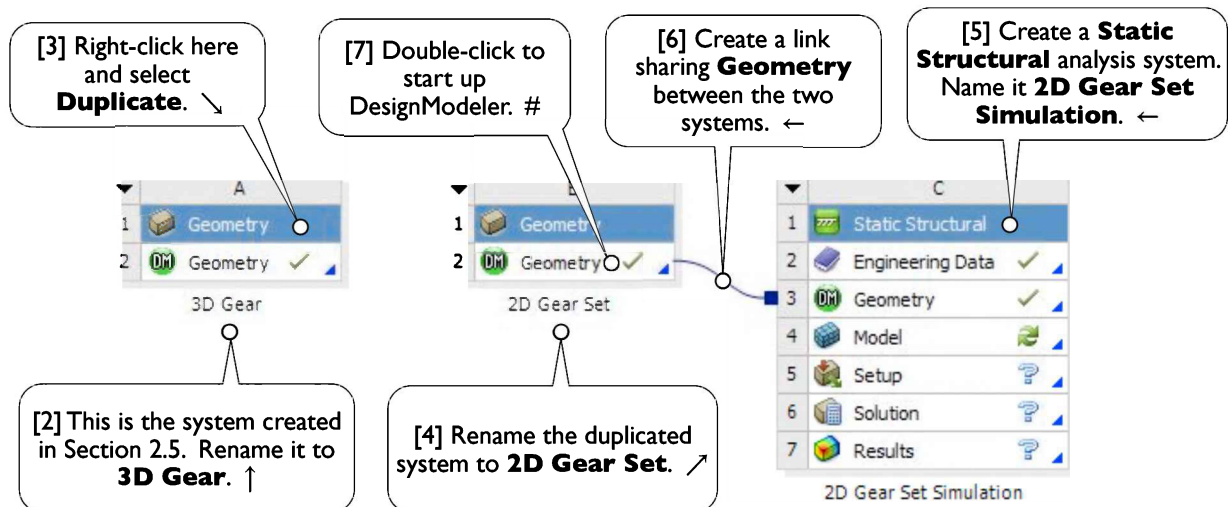
[1] In this section, we'll perform a simulation for a pair of meshing spur gears introduced in Section 2.5. Consider a situation in which a clockwise torque of 15,000 lb-in is applied on the upper gear while both gears are at rest. Our goal is to assess the maximum stress during the transmission of the torque. By engineering judgement, the maximum stress occurs either at a contact point [2] or at the root of a tooth due to the bending of the tooth [3].

Since there is no restriction of deformation in the depth direction, i.e., the gears are free to expand (or contract) in the depth direction, so it is modeled as a plane stress problem (3.3.1, page 140). The U.S. customary unit system (**in-lbm-lbf-s**) is used in this exercise. →



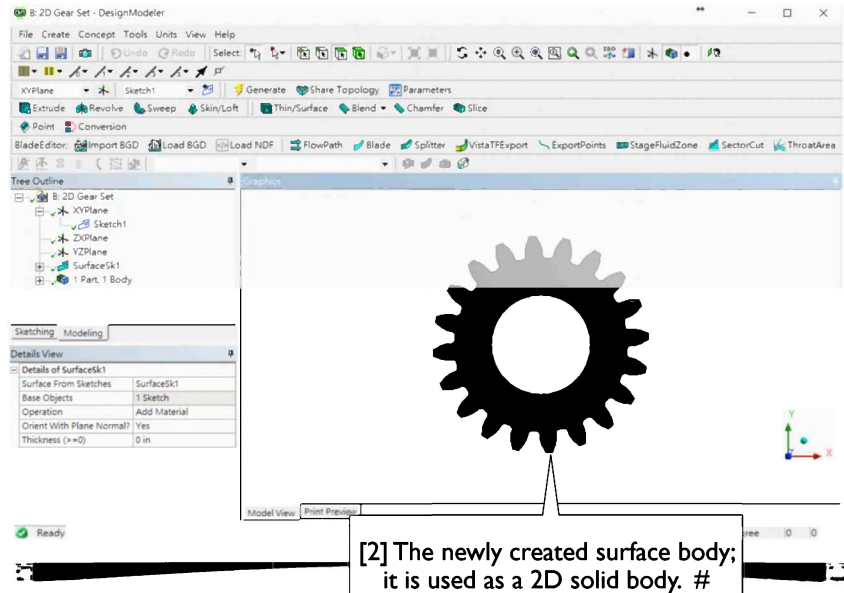
3.4.2 Set Up Project Schematic

[1] Launch Workbench. Open the project **Gear**, which was saved in Section 2.5 (see [2]). Duplicate the **Geometry** system [3-4]. Create a **Static Structural** system [5]. Create a link between the last two systems so that they share **Geometry** [6]. Double-click the **Geometry** cell in the **2D Gear Set** system to edit it [7]. ✓



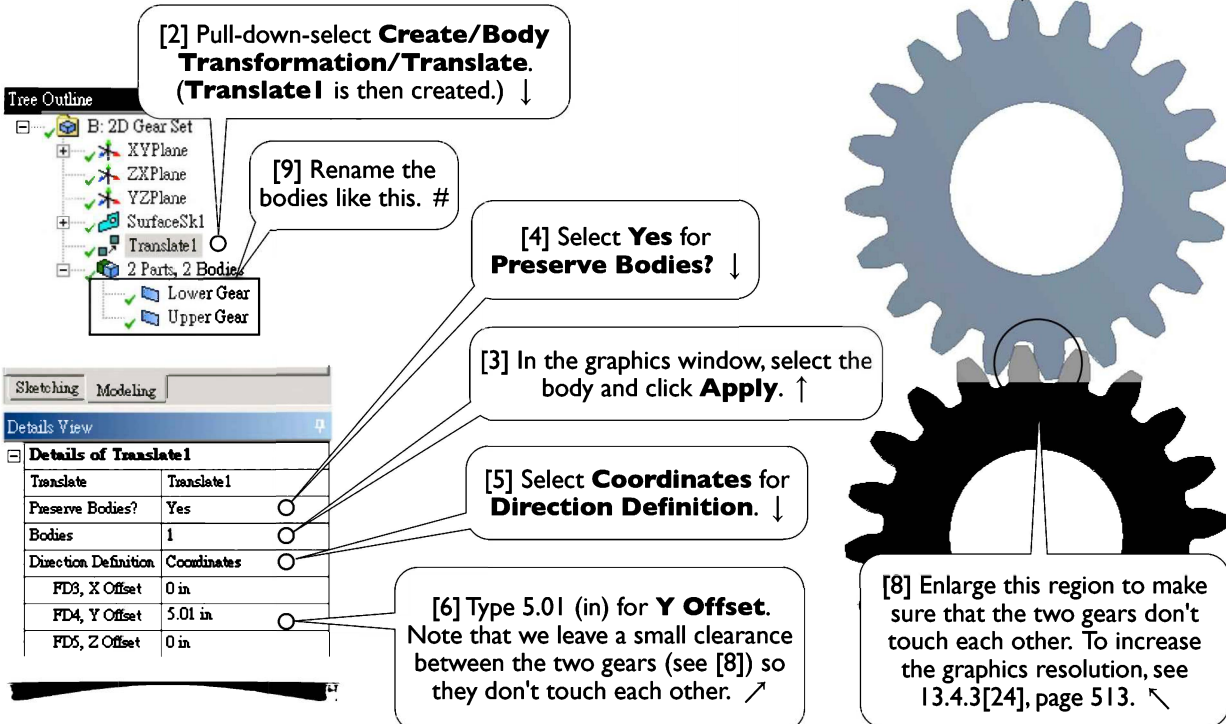
3.4.3 Delete the 3D Body and Create a Surface Body

[1] Delete **Extrude1**. Create a surface body using **Sketch1** by pull-down-selecting **Concept/Surfaces From Sketches**. Remember to click **Generate**. →



3.4.4 Duplicate the Gear

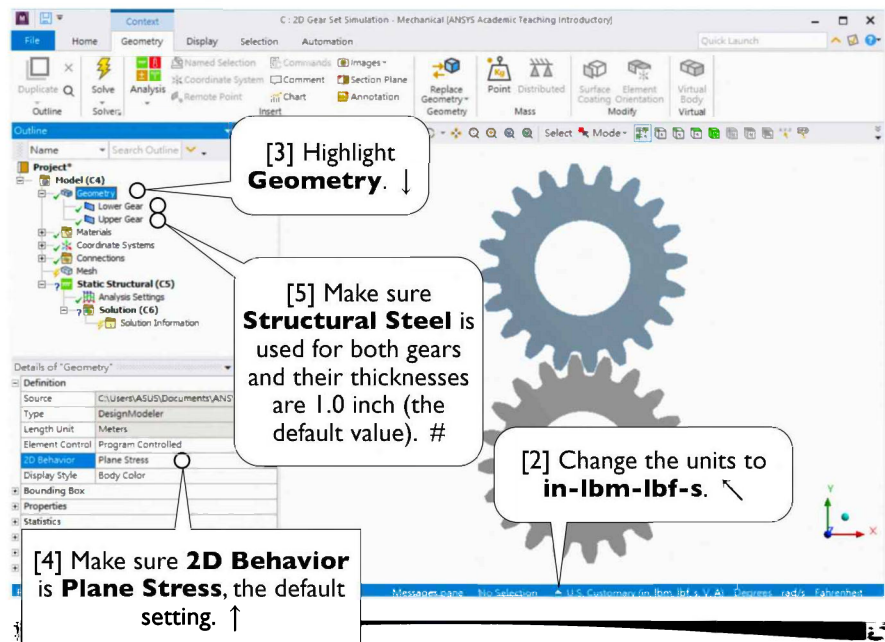
[1] Duplicate the gear [2-8]. Rename the two bodies to **Lower Gear** and **Upper Gear**, respectively [9]. Close DesignModeler. ↓



3.4.5 Set Up Geometry in Mechanical

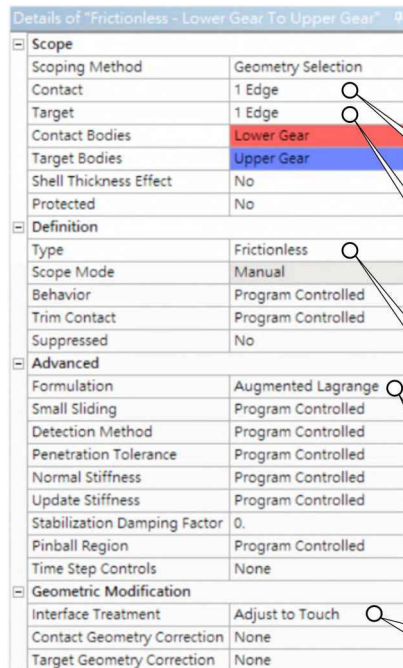
[1] Before entering **Mechanical**, specify **2D** for **Analysis Type** (3.1.4[4-6], page 112). This step is important since, after the geometry is brought to **Mechanical**, you cannot change it any more.

Start up **Mechanical** by double-clicking the **Model** cell. In **Mechanical**, change the units to **in-lbm-lbf-s** [2]. Make sure **2D Behavior** is set to **Plane Stress** [3-4]. Also make sure the material is **Structural Steel** (default) and the thickness is 1.0 inch (default) [5]. →



3.4.6 Set Up Contact Region

[1] Redefine the contact region as follows. In the project tree, highlight **Connections/Contacts/Contact Region** and redefine **Contact** and **Target** [2-5]. Change the contact type to **Frictionless** [6]. Select **Augmented Lagrange** for **Formulation** [7]; this is generally recommended for frictionless contact (13.1.10, pages 477-478). Finally, select **Adjust to Touch** for **Interface Treatment** [8] (13.1.11, page 479). ↓



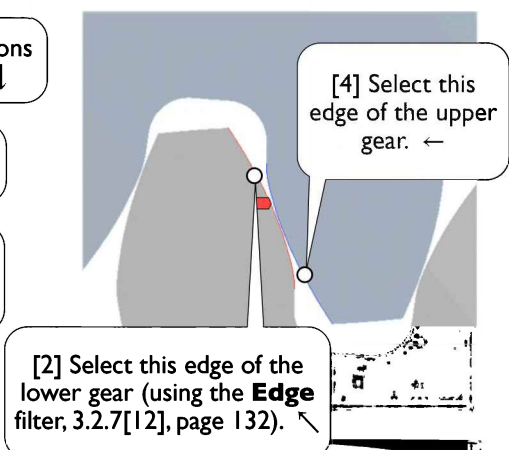
[3] Click to bring up buttons and apply as **Contact**. ↓

[5] Click to bring up buttons and apply as **Target**. ↓

[6] Select **Frictionless** for **Type**. ↓

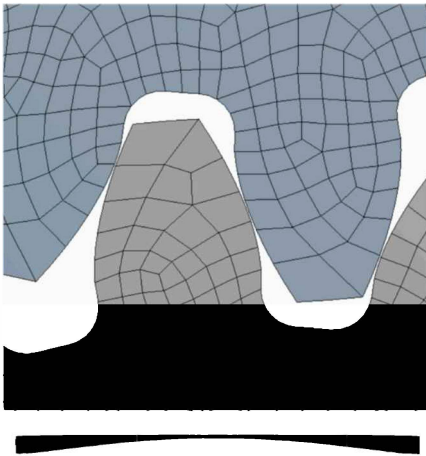
[7] Select **Augmented Lagrange** for **Formulation**. ↓

[8] Select **Adjust to Touch** for **Interface Treatment**. #

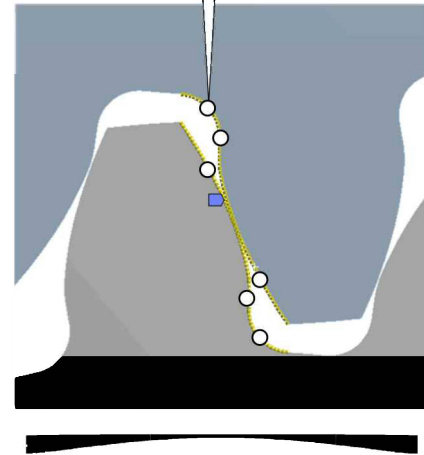


3.4.7 Generate Mesh

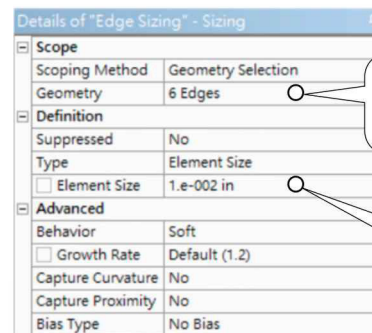
[1] Generate mesh [2]. The mesh with default settings is too coarse. We need finer mesh on the contact areas and the fillets. Insert a **Sizing** for the mesh [3-5]. The new mesh should be adequate now [6-8]. ↓



[2] Mesh with default settings. →

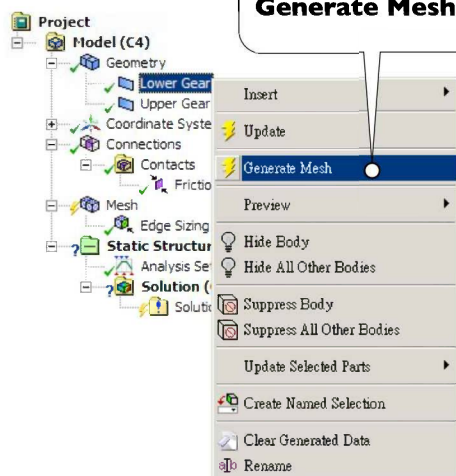


[3] Insert a **Sizing** (3.2.8[7], page 134). Control-select these 6 edges. ↓

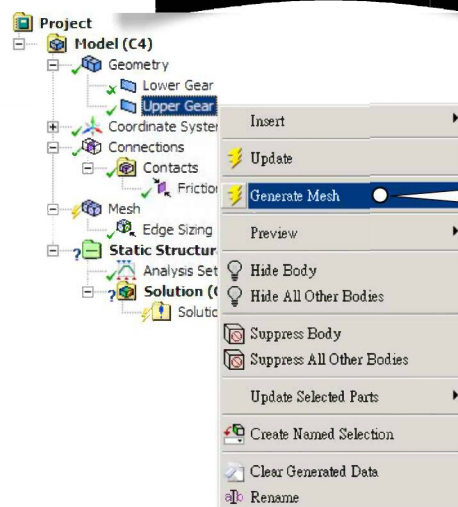


[4] And click **Apply**. ↓

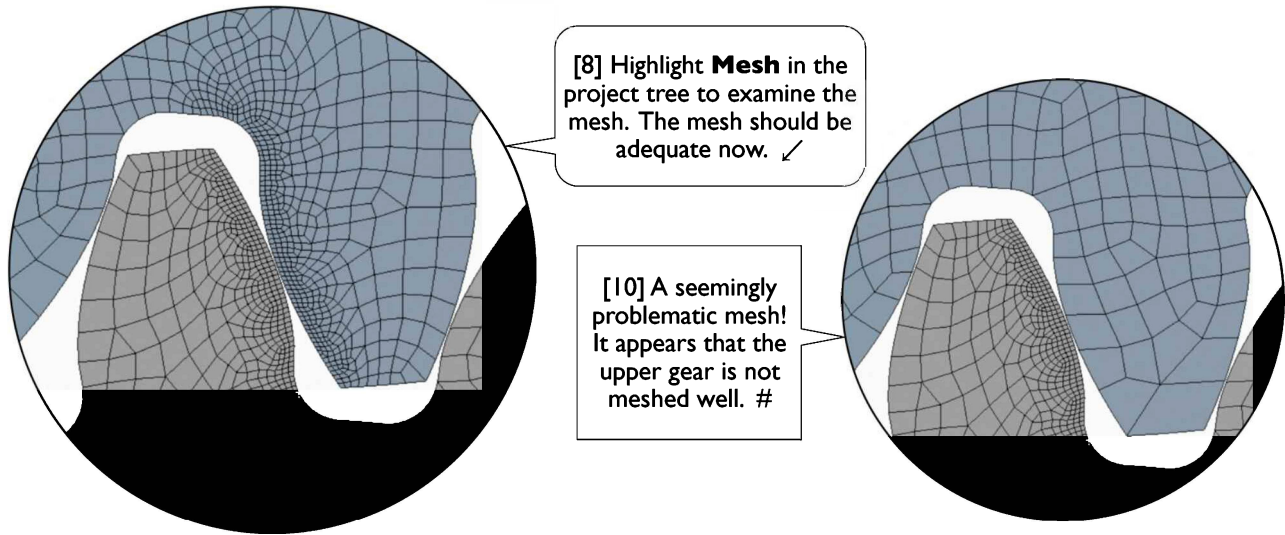
[5] Type 0.01 (in) for **Element Size**. ←



[6] Right-click **Lower Gear** and select **Generate Mesh**. ↘



[7] Right-click **Upper Gear** and select **Generate Mesh**. ↙



[9] Here, we've demonstrated that each body can be meshed separately. Another reason we meshed this way is to circumvent an insignificant bug in Workbench. If you right-click **Mesh** and issue **Generate Mesh** as usual (instead of [6-7]), you would have a seemingly problematic mesh [10]. The mesh is actually fine; it is just a display problem. ↑

3.4.8 Specify Support and Load

Static Structural
Time: 1. s

- A Fixed Support
- B Frictionless Support
- C Moment: -15000 lbf·in

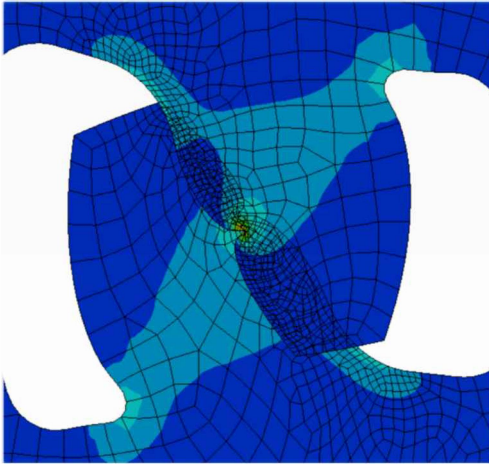
[1] Apply a **Fixed Support** on the inner rim of the lower gear. →

[2] Apply a **Frictionless Support** on the inner rim of the upper gear. This restricts the inner rim from radial translation, while allowing tangential rotation. ↑

[3] Apply a **Moment** of 15,000 lbf·in clockwise on the inner rim of the upper gear. (Type -15000 in the details view.) This is the driving torque. #

Details of "Moment"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Type	Moment
Magnitude	-15000 lbf·in (ramped)
Suppressed	No
Behavior	Deformable
Advanced	

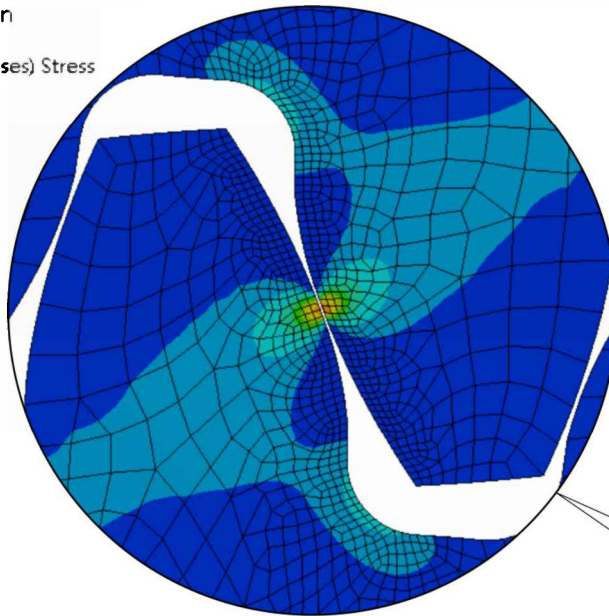
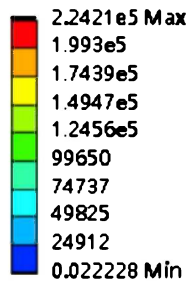
3.4.9 Solve the Model and View the Results



[1] Insert an **Equivalent (von-Mises) Stress**, solve the model, and view the results. With the default scale (**Auto Scale**), the deformation is exaggerated (140 times) and looks unrealistic. ↓

1.4e+002 (Auto Scale) ↓
Scoped Bodies
☒ Large Vertex Contours

C: 2D Gear Set Simulation
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: psi
Time: 1



[2] Change the deformation scale to **True Scale**. ↓

1.0 (True Scale) ↓
Scoped Bodies
☒ Large Vertex Contours

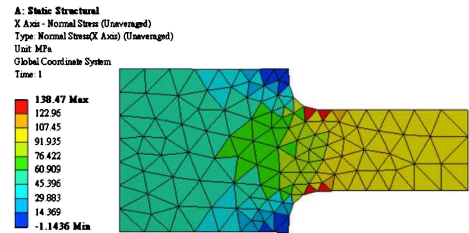
[3] Now, examine the contact stress and the bending stress. Note that the unit for stress is psi. ↓

Wrap Up

[4] Close **Mechanical**, save the project, and exit Workbench. #

Section 3.5

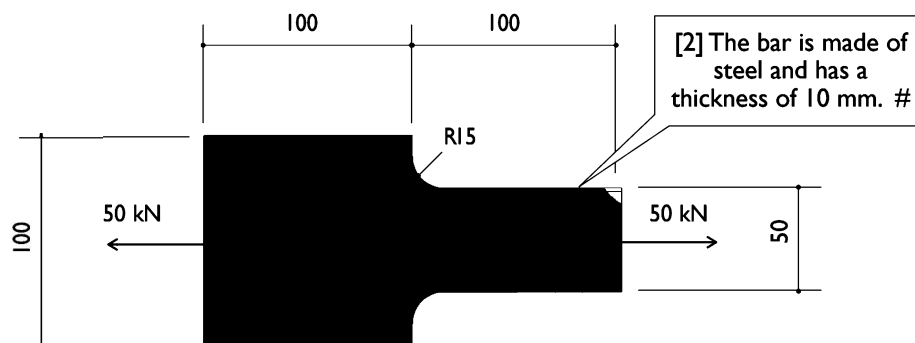
Structural Error, FE Convergence, and Stress Singularity



This exercise illustrates some must-know concepts in finite element simulations: (a) stress discontinuity, (b) structural error, (c) finite element convergence, (d) stress concentration, and (e) stress singularity. We use a filleted bar subject to tensile stresses to demonstrate these concepts.

3.5.1 About the Filleted Bar

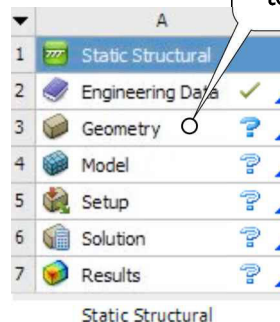
[1] The filleted bar is made of steel and subject to a tension of 50 kN as shown in [2]. We want to investigate the maximum displacement and the maximum normal stress in horizontal direction. The maximum normal stress occurs near the fillets where the stress concentration occurs. The unit system used in this exercise is **mm-kg-N-s**. ↓



PART A. STRESS DISCONTINUITY

3.5.2 Start a New Project

[1] Launch Workbench. Create a **Static Structural** system. Save the project as **Bar**. Start up DesignModeler [2]. →

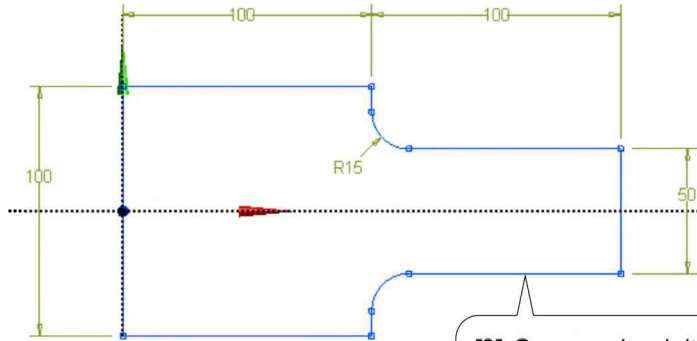


[2] Right-click **Geometry** cell and select **New DesignModeler Geometry...** to start up DesignModeler. #

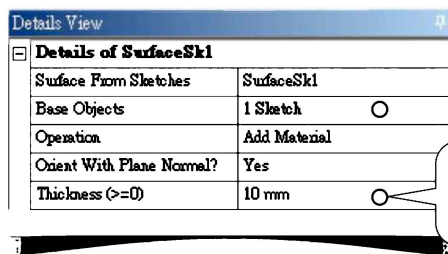
3.5.3 Create a 2D Model in DesignModeler

[1] Select **Millimeter** as the length unit. Create a sketch on **XYPlane** shown in [2]; remember to impose **Symmetry** constraints. Create a surface body by pull-down-selecting **Concept/Surfaces from Sketches** [3-4]. Close DesignModeler.

In the **Project Schematic**, specify **2D** for **Analysis Type** (3.1.4[4-6], page 112). Start up **Mechanical** by double-clicking **Model**. →



[2] Create a sketch like this on **XYPlane**. ✓



[3] Type 10 (mm) for thickness. →



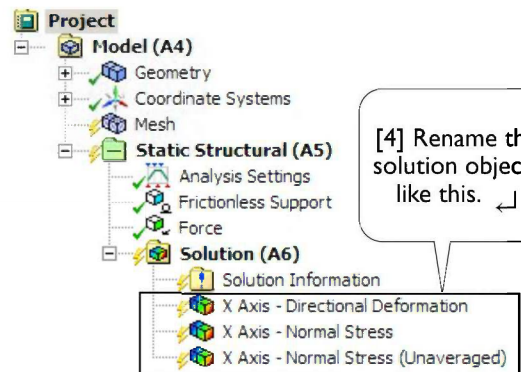
[4] Click **Generate** to create this surface body (2D solid body). #

3.5.4 Set Up Support, Load, and Solution Objects

[1] In **Mechanical**, select **mm-kg-N-s** unit system.

Specify a **Frictionless Support** on the left edge [2], and a horizontal force of 50,000 N on the right edge [3].

Under the **Solution** branch, insert a **Directional Deformation** and two **Normal Stress**. Rename the objects as shown in [4]. In the details of **X Axis - Normal Stress (Unaveraged)**, select **Unaveraged** for **Display Option** ([5], next page); the significance of this setting will be explained in 3.5.6, page 160. ↓



[4] Rename the solution objects like this. ↩

A: Static Structural
Static Structural
Time: 1. s

A Frictionless Support
B Force: 50000 N

[2] Specify a **Frictionless support** on this edge. →



[3] Specify a horizontal force of 50,000 N on this edge. ↑

Details of "X Axis - Directional Deformation"	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
[-] Definition	
Type	Directional Deformation
Orientation	X Axis
By	Time
<input type="checkbox"/> Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
[+] Results	

Details of "X Axis - Normal Stress"	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
[-] Definition	
Type	Normal Stress
Orientation	X Axis
By	Time
<input type="checkbox"/> Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
[-] Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
[+] Results	

Details of "X Axis - Normal Stress (Unaveraged)"	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
[-] Definition	
Type	Normal Stress
Orientation	X Axis
By	Time
<input type="checkbox"/> Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
[-] Integration Point Results	
Display Option	Unaveraged
[+] Results	

[5] Select **Unaveraged** for **Display Option**. #

3.5.5 Set Up Mesh Controls

[1] Highlight **Mesh** in the project tree and, in the details view, select **Linear** for **Element Order** [2]. This sets to the use of lower-order elements (see 1.3.2[2-4], page 37).

Right-click **Mesh** and select **Insert/Method** (or select **Mesh/Controls/Method** from the toolbars) and select the body as **Geometry** [3]. Select **Triangles** for **Method** [4]. This sets to the use of triangular elements (see 1.3.3[6], page 38).

Generate the mesh [5]. Turn on **Weak Springs** (3.1.8[1-2], page 116) and click **Solve** to solve the model, ignoring the warning messages.

Why Lower-Order Triangular Elements?

The purpose of **PART A** is to demonstrate *stress discontinuity*, a must-know in finite element solutions. We've set up a mesh that uses lower-order (linear) triangular elements. The purpose of this setup is to exaggerate the stress discontinuity behavior. Stress discontinuity is intrinsic in all the finite element software using displacements as degrees of freedom, and is not limited to triangular or lower-order elements. →

Details of "Mesh"	
[-] Display	
Display Style	Use Geometry Setting
[-] Defaults	
Physics Preference	Mechanical
Element Order	Linear
<input type="checkbox"/> Element Size	Default (15.359 mm)
[+] Sizing	
[+] Quality	
[+] Inflation	
[+] Advanced	
[+] Statistics	

[2] Select **Linear** for **Element Order**. ✓

Details of "All Triangles Method" - Method	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
[-] Definition	
Suppressed	No
Method	Triangles
Element Midside Nodes	Use Global Setting

[3] Select the body. ↓

[4] Select **Triangles** for **Method**. →

[5] The mesh. #

3.5.6 View the Results

Displacement Fields

[1] Nodal displacements, calculated by solving Eq. 1.3.1(1) (page 35), are single valued (each node has a single value). Therefore the displacement fields, calculated using Eq. 1.3.2(2) (page 36), are continuous over the entire structural body [2].

The displacement fields are continuous but not necessarily smooth. The use of continuous shape functions within an element guarantees the displacements field is piecewise smooth, but not necessarily smooth across the element boundaries.

Stress Fields

The strain fields are then calculated using Eq. 1.2.7(1) (page 31), and stress fields are calculated, element by element, using Eq. 1.2.8(1) (page 31). The figure in [3-4] is a typical result of stress calculation: a node usually has multiple stress values, since the node may connect to multiple elements, and each element has its own stress value.

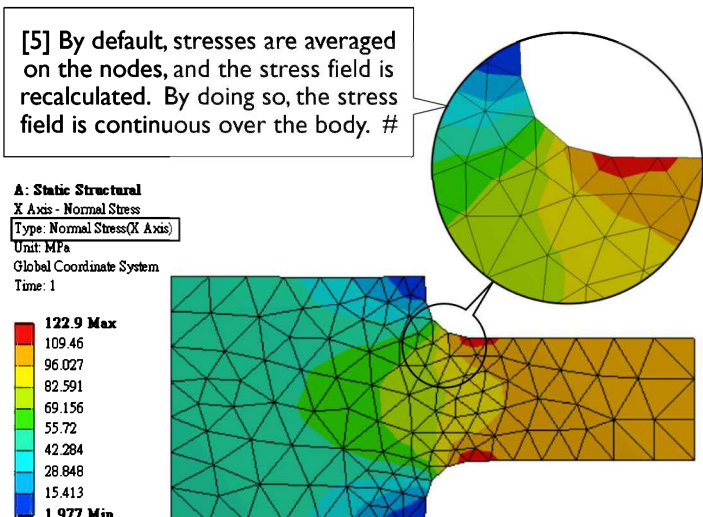
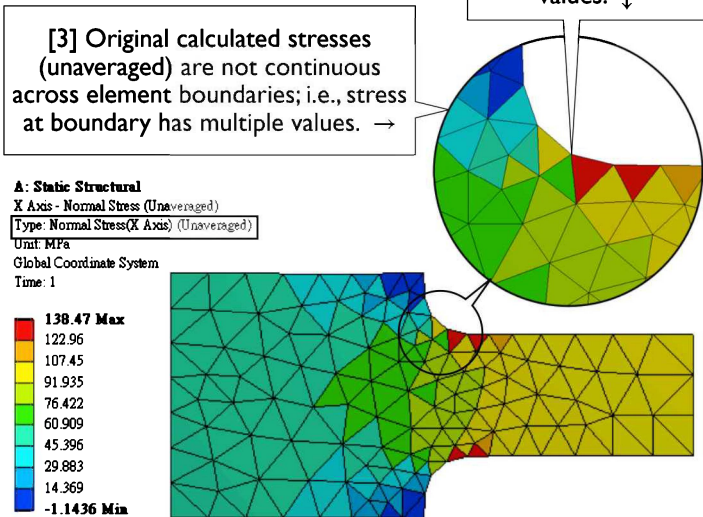
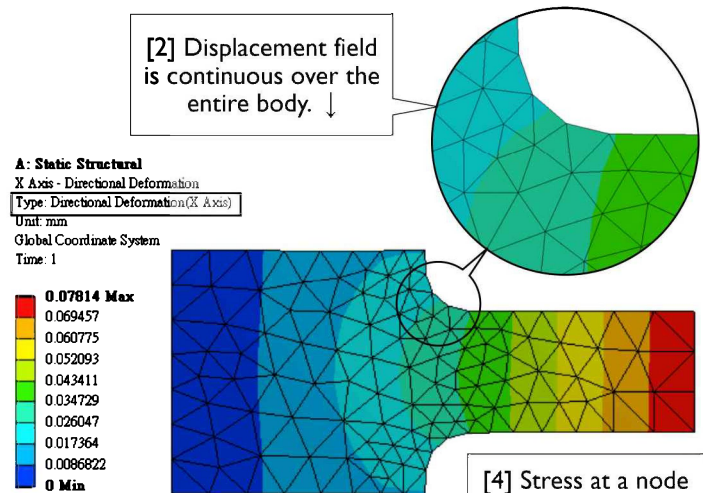
This behavior can be easily understood. Since Eq. 1.2.7(1) involves differentiations of piecewise smooth displacement fields, this ensures the strain fields and the stress fields are continuous inside the element but not necessarily continuous across the element boundaries.

By default, stresses are averaged first on the nodes, and the stress fields are in turn recalculated. By doing so, the stress field is continuous over the body as shown in [5].

Usage of Unaveraged Stresses

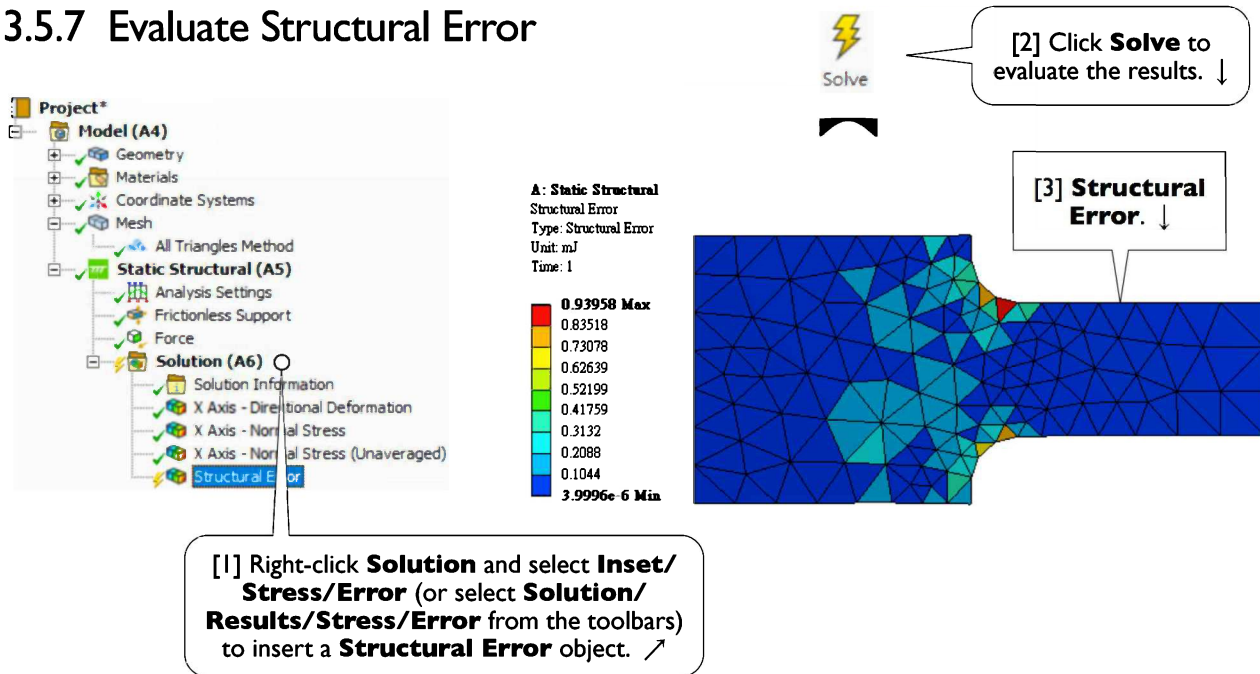
The averaged stress fields [5] are visually efficient for human eyes to interpret the results, while unaveraged stress fields [3-4] provide a way of assessing the solution accuracy.

In general, as the mesh is getting finer, the solution is more accurate, and the stress discontinuity is less obvious. Thus, stress discontinuity can be used as an indicator for the solution accuracy: the less discontinuous the stress field, the more accurate the solution. ↗



PART B. STRUCTURAL ERROR

3.5.7 Evaluate Structural Error



Structural Error^[Ref 1]

[4] For an element, strain energy calculated using averaged stresses are different from that using unaveraged stresses. The difference between the two strain energy values is called **Structural Error** of the element. The finer the mesh, the smaller the structural error.

The structural error can be used for two purposes: (a) As an indicator of global mesh adequacy. In general, we want the values as small as possible. Refining the mesh globally is a way of reducing structural error. (b) As an indicator of the local mesh adequacy. In general, we want the structural error distribution as uniform as possible to optimize the efficiency of computation effort. That is, in the region of large values of structural error we should reduce the element size, while in the region of small values of structural error we should enlarge the element size. #

PART C. FINITE ELEMENT CONVERGENCE

One of the core concepts of the finite element methods is that *the finer the mesh, the more accurate the solution*. Ultimately, the solution will reach an analytical solution as the mesh is fine enough. But, how fast does it approach the analytical solution? This is what we want to answer in this part of the section. It is called the *element convergence behavior*.

The answer depends on what kind of element we are using. We'll draw a conclusion that, for 2D cases, *quadrilateral elements generally converge faster than triangular elements*.

We will compare lower-order triangular with lower-order quadrilateral. To be fair, the comparison is made under the same problem sizes for both types of elements. The problem size is reasonably defined by the number of nodes.

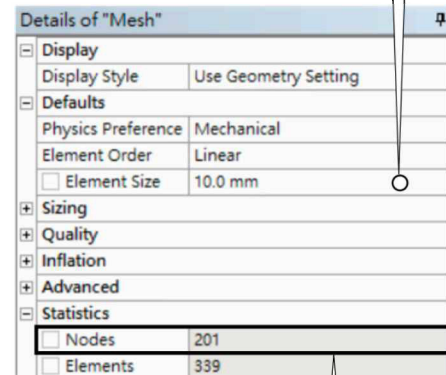
As a representative structural response, the maximum horizontal displacement is used for comparison.

3.5.8 Triangular Elements

[1] Repeat the simulation as follows: Change the element size as shown in the table below for each run [2], record the number of nodes [3], and the maximum directional deformation (displacement). Tabulate the results like this: →

Element Size (mm)	Number of Nodes	Max Displacement (mm)
10	201	0.078113
8	295	0.078034
6	490	0.078262
5	698	0.078380
4	1066	0.078454
3	1857	0.078545
2.5	2688	0.078575
2	4109	0.078603
1.5	7238	0.078626
1.2	11050	0.078635
1	15962	0.078640

[2] Type the element size for each run. This sets to a uniform-sized mesh. ↓



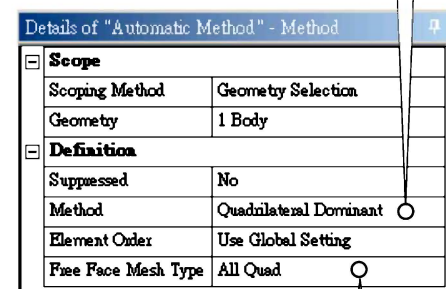
[3] Number of nodes. #

3.5.9 Quadrilateral Elements

[1] Highlight **All Triangles Method** (which is under the **Mesh** branch). In the details view, change to **Quadrilateral Dominant** for **Method** [2], and select **All Quad** for **Free Face Mesh Type** [3]. Repeat the steps in 3.5.8, and tabulate the results like this: →

Element Size (mm)	Number of Nodes	Max Displacement (mm)
10	196	0.078329
8	293	0.078325
6	484	0.078463
5	689	0.078529
4	1063	0.078557
3	1803	0.078609
2.5	2630	0.078619
2	4043	0.078631
1.5	6966	0.078641
1.2	11018	0.078646
1	15747	0.078647

[2] Change to **Quadrilateral Dominant**. ↓



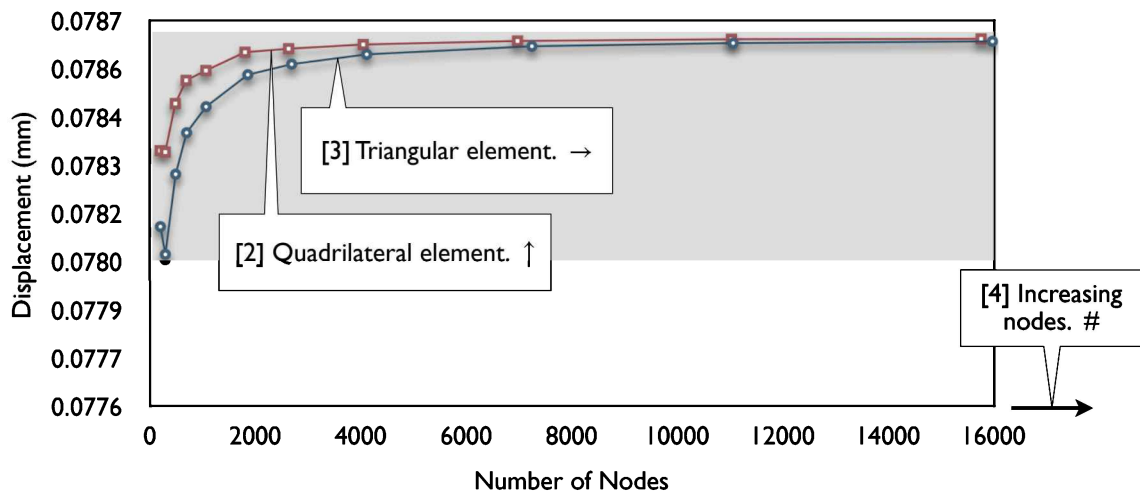
[3] Select **All Quad**. #

3.5.10 Comparison and Conclusions

[1] Plotting the results in 3.5.8 and 3.5.9 (last page) using a spreadsheet program such as Microsoft Excel, you should come up with a chart as shown below [2-4]. The two curves share the same horizontal asymptote (Displacement ≈ 0.07864 mm), which is the analytical solution. Additional behaviors can be observed as follows.

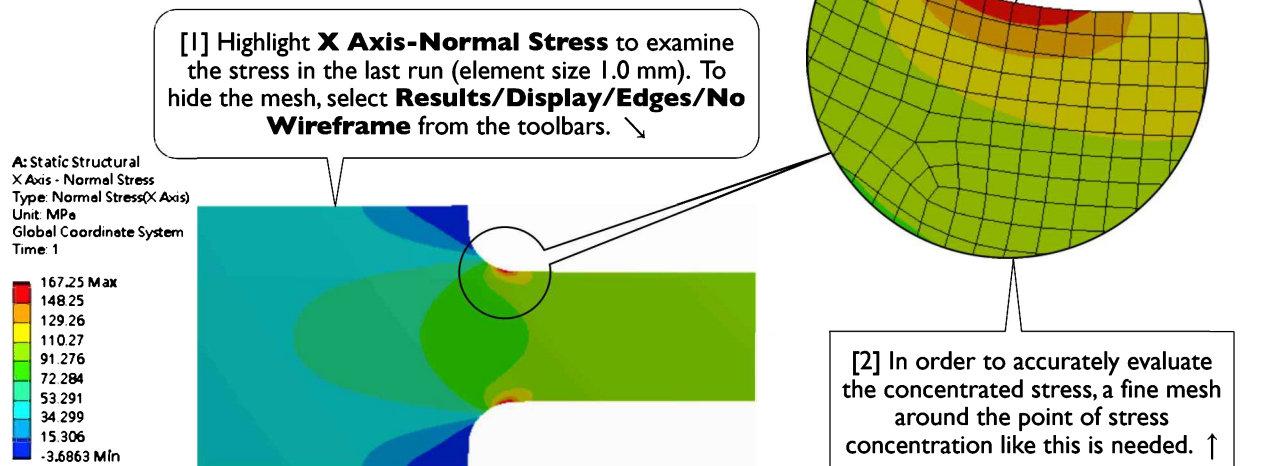
First, the quadrilateral element converges to the analytical solution faster than the triangular element. The difference seems undramatic in this particular case. However, in many other cases, the difference can be significant. In Section 9.3 we will investigate the convergence behavior of 3D elements, and we'll conclude that the convergence rate of the hexahedra (1.3.3[2], page 38) is faster than the prisms or the tetrahedra (1.3.3[3, 5], page 38).

Second, all the convergence curves approach the asymptote from below; i.e., displacements calculated by the finite element methods never exceed the analytical solution. In other words, the finite element solutions always underestimate the deformation, or in terms of stiffness (1.3.1[4], page 36), the stiffness is always overestimated. ✓



PART D. STRESS CONCENTRATION

3.5.11 View the Stress



3.5.12 Define a Path

[1] Right-click **Model** and select **Insert/Construction Geometry/Path** (or select **Model/Prepare/Construction Geometry/Path** from the toolbars) ↓

[2] A path with the name **Path** is inserted. →

[4] Click **Apply.** ↗

[6] Click **Apply.** →

Details of "Path"	
Definition	
Path Type	Two Points
Path Coordinate System	Global Coordinate System
Number of Sampling Points	47
Suppressed	No
Start	
Coordinate System	Global Coordinate System
Start X Coordinate	115. mm
Start Y Coordinate	25. mm
Location	Click to Change
End	
Coordinate System	Global Coordinate System
End X Coordinate	115. mm
End Y Coordinate	-25. mm
Location	Click to Change

[3] Select this vertex. You may need to turn on the **Vertex** filter. ✓

[5] Select this vertex. ✓

What are we doing?

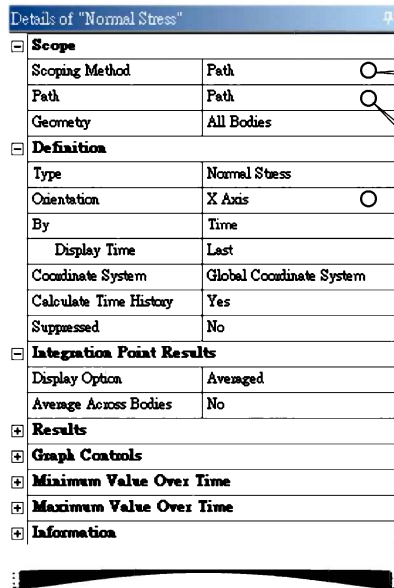
[7] We are creating a path. We'll investigate stresses along this path, to see how the stresses vary with the path.

A path can be defined by two points (in that case, it is a straight path), or an edge (in that case, possibly a curve path). Coordinates of the points can be either picked from the model or typed in the details view.

Any result objects can scope with a path. After solving, a result-versus-path data table along with a graph will be generated.

When a path is short enough, it essentially becomes a single point. It is useful when you want to investigate the results of a specific point. #

3.5.13 View Stresses Along the Path



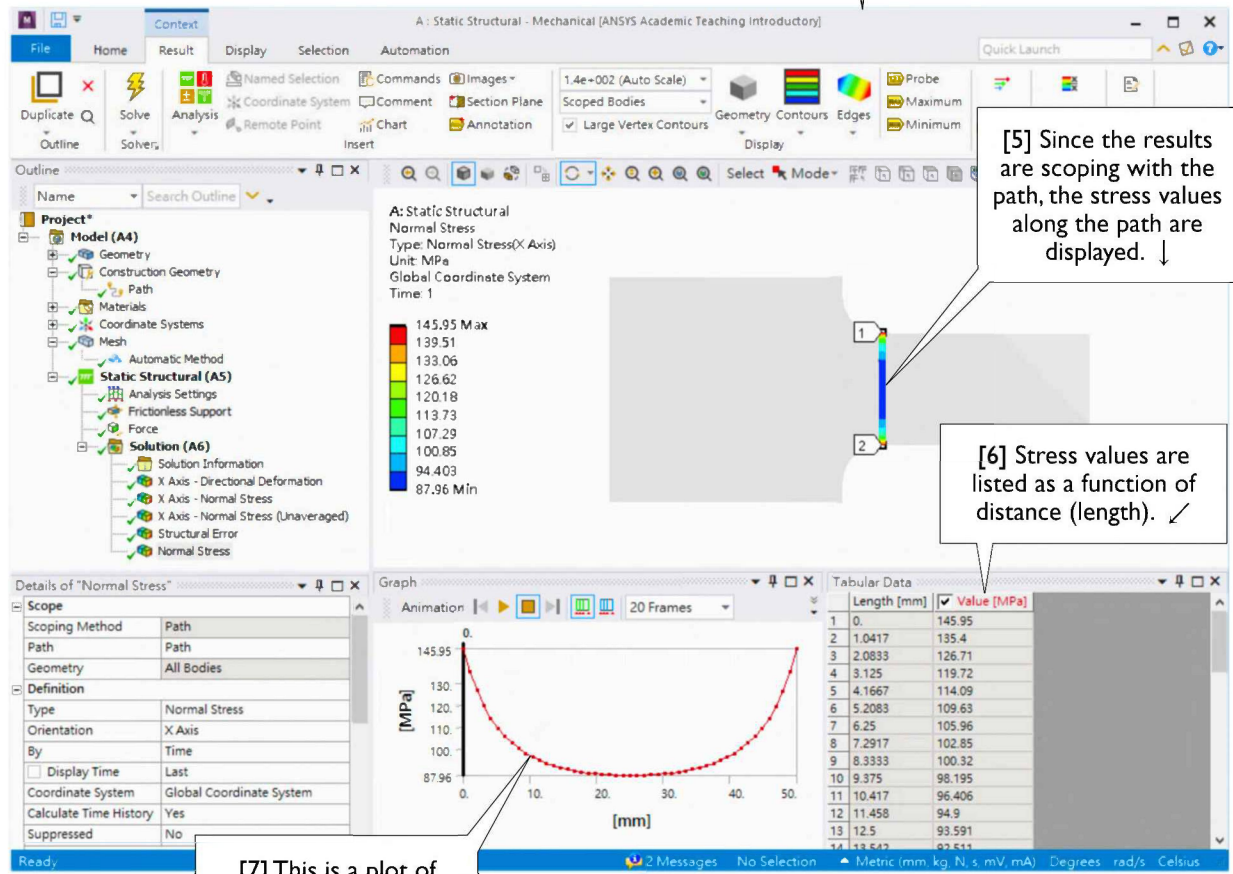
[1] Insert a **Normal Stress** object and, in the details view, select **Path** as **Scoping Method**. ↓

[2] Pull-down-select **Path**, which is the name of the newly created path. →



[3] Solve. ↓

[4] With **Normal Stress** highlighted, **Mechanical GUI** looks like this. ↓



[5] Since the results are scoping with the path, the stress values along the path are displayed. ↓

[6] Stress values are listed as a function of distance (length). ✓

[7] This is a plot of stress-versus-distance. #

PART E. STRESS SINGULARITY

Many engineers are aware of the concept of stress concentration: stress is larger at a concave corner than stress at locations away from the corner. To accurately evaluate the concentrated stress, finer mesh needs to be used around the corner. As a general guideline, *an area of higher stress gradient requires finer mesh*.

The degree of stress concentration is related to the radius of the corner: the smaller the radius, the larger the concentrated stress. Naturally, a question comes up: what happens if the radius of the concave corner is zero (i.e., a sharp angle corner)? The elasticity theory predicts that, in that case, the stress at that sharp corner is infinity. A stress of infinity is called a *singular stress*.

Fortunately, a corner with zero radius never exists in the real-world. It is difficult to manufacture such a zero-radius fillet. It requires a process of infinite-high precision machining.

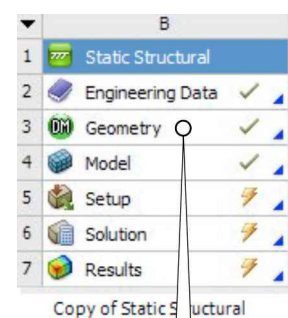
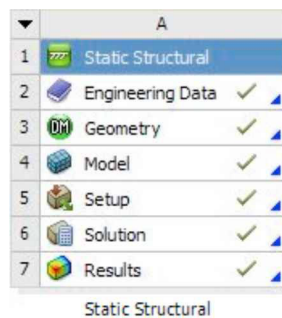
However, zero-radius fillets do exist in the virtual-world of simulations. Since many small features such as fillets do not significantly affect the global behavior (e.g., deformation) of a structure, these small features are often not modeled in the simulation model. The consequence is that zero-radius fillets exist everywhere in a simulation model.

It is important that the engineers be aware of the existence of stress singularities. Novice engineers often mistakenly take the maximum stress as the design stress, while, in fact, that is a singular stress—it doesn't exist in the real-world. Always check if the maximum stress occurs at a singular point. If so, the stress is meaningless. If a concentrated stress is important, always include the fillet in the model.

3.5.14 Modify the Geometric Model

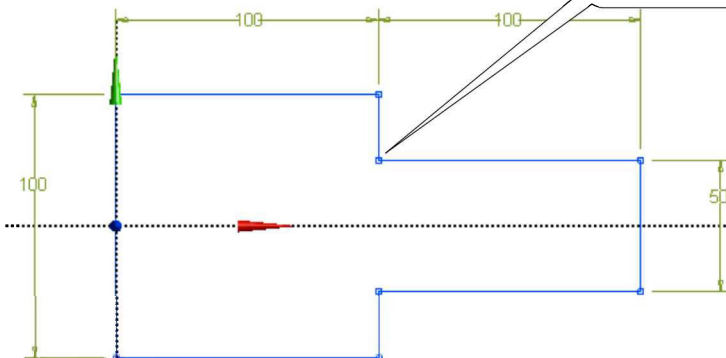
[1] Close **Mechanical**. Duplicate the **Static Structural** system. Double-click **Geometry** of the new system [2].

Modify the sketch such that the fillets become sharp corners [3]. Click **Generate** to update the surface body. Reduce the model to half by taking advantage of symmetry [4] (using **Tools/Symmetry**). We reduce the model because we want to use very fine mesh to study stress singularity. Close DesignModeler and open **Mechanical** by double-clicking **Model** in the new system (click **Yes** to read the upstream data). →



[2] Double-click **Geometry**. ←

[3] Modify the sketch such that the fillets become sharp corners. ↓

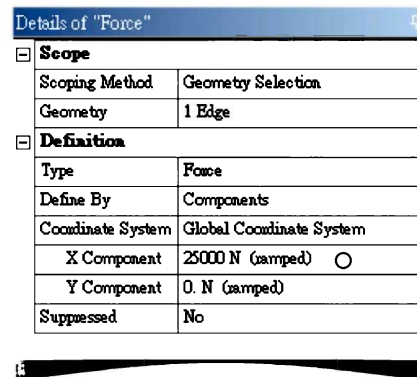


[4] Use **ZXPlane** as the plane of symmetry. #

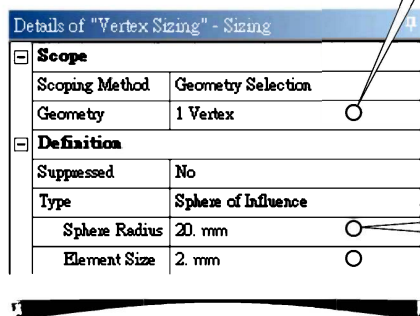
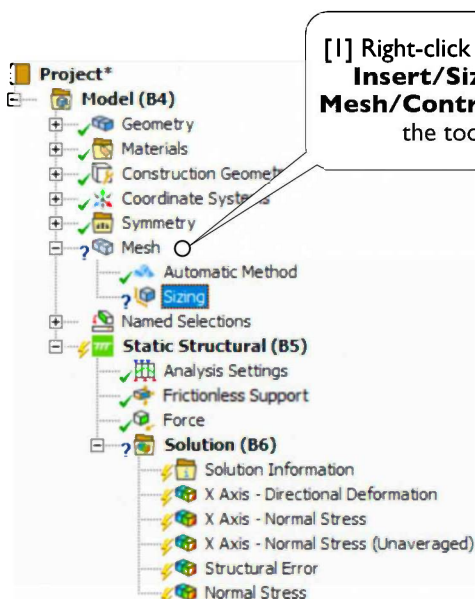
3.5.15 Modify the Environment Conditions in **Mechanical**

B: Copy of Static Structural
Force
Time: 1. s

Force: 25000 N
Components: 25000, 0. N

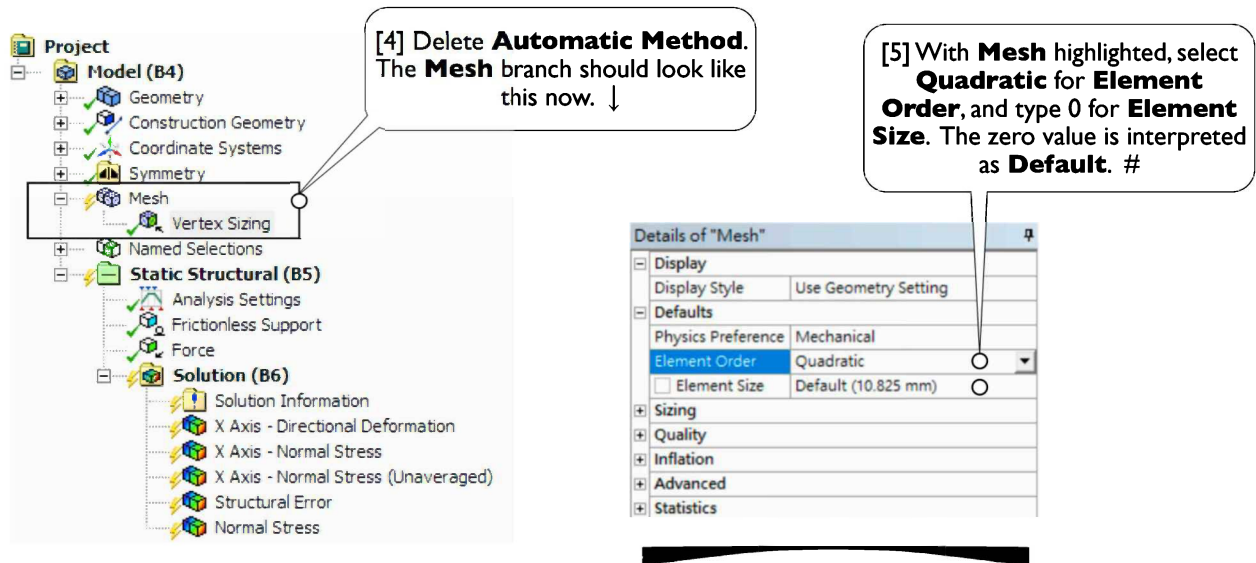


3.5.16 Set Up Mesh Controls

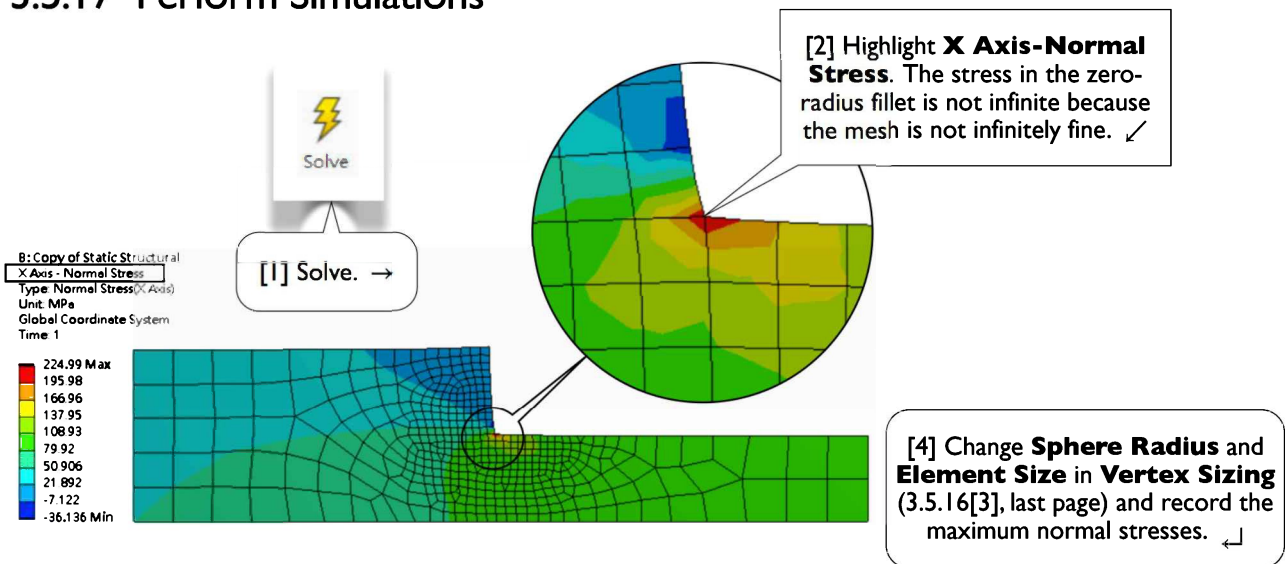


[2] Select the vertex at the concave sharp corner and click **Apply**. You may need to turn on **Vertex** filter. ↓

[3] Type 20 (mm) for **Sphere Radius** and 2 (mm) for **Element Size**. For the following study, we will change **Element Size** and use 10 times the element size or at least 3 mm as **Sphere Radius** of influence (3.5.17[3-4], next page). ↩



3.5.17 Perform Simulations

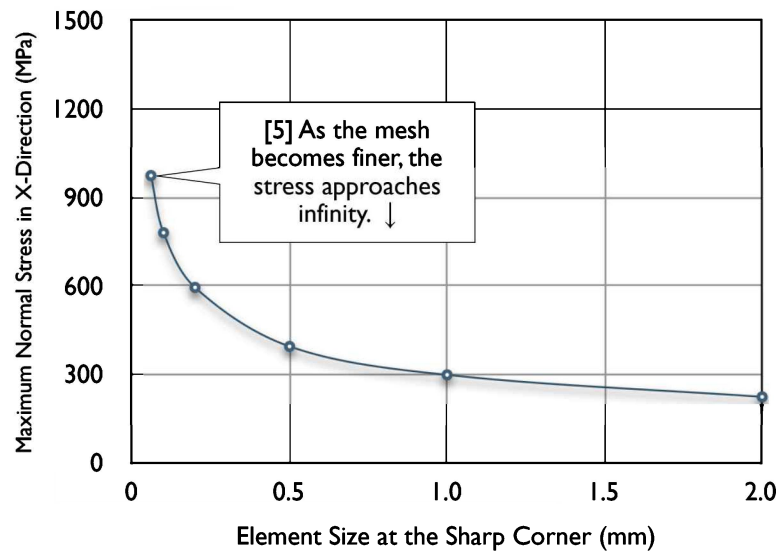


[3] After solving the model, the stress at the zero-radius fillet is not infinite, the theoretical value. This is because our mesh is not fine enough. To achieve an infinite value of stress, you need a zero element size. That is, of course, not possible.

Reduce **Sphere Radius** and **Element Size** in **Vertex Sizing** (3.5.16[3], last page) and record the maximum stress. Repeat this and you should obtain a table like [4].

Interpretation of the data may be easier if we plot them in a graph ([5], next page). The graph reveals that, as the mesh gets finer (from right to left), the stress at the sharp corner eventually reaches an infinite value, consistent with the theoretical value. ↗

Element Size (mm)	Sphere Radius (mm)	Max Normal Stress (X Axis) (MPa)
2	20	224.99
1	10	299.34
0.5	5	395.52
0.2	3	594.84
0.1	3	779.80
0.06	3	973.13



Remark

[6] Stress singularity is not limited to concave sharp corners. Any locations that have stress of infinity are called singular points. For example, a point of concentrated forces is also a singular point, since a point has zero area. ↓

Wrap Up

[7] Save the project. Close **Mechanical** and exit Workbench. #

Reference

1. All Help>Mechanical APDL>Theory Reference>17.6.1. Error Approximation Technique for Displacement-Based Problems

Section 3.6

Review

3.6.1 Keywords

Choose a letter for each keyword, from the list of descriptions

- | | |
|------------------------------------|-------------------------------|
| 1. () APDL | 7. () Project Tree |
| 2. () Element Convergence Study | 8. () Stress Discontinuity |
| 3. () Environment Conditions | 9. () Stress Singularity |
| 4. () Inconsistent Unit System | 10. () Structural Error |
| 5. () Plane Strain Problem | 11. () Weak Spring |
| 6. () Plane Stress Problem | |

Answers:

1. (F) 2. (J) 3. (G) 4. (E) 5. (C) 6. (B) 7. (D) 8. (H)
9. (K) 10. (I) 11. (A)

List of Descriptions

(A) When Workbench detects a structure as unstable, it adds weak springs on the structure to make it capable of withstanding very small external forces.

(B) In a structural simulation problem, if all the stresses in a direction, say Z-direction, vanish, the problem can be reduced to a 2D problem.

(C) In a structural simulation problem, if all the strains in a direction, say Z-direction, vanish, the problem can be reduced to a 2D problem.

(D) A structured representation of an analysis system and displayed on the **Outline** in **Mechanical**. It contains one or more simulation models. A simulation model contains one or more **Environment** branches.

(E) A unit system in which at least one unit is not consistent with other units. For example, **mm-kg-N-s** is inconsistent, since if mm and kg is used, the force must be mN (milli-newton) instead of N.

- (F) ANSYS Parametric Design Language. A set of text-based language that is used to drive ANSYS Classic program.
- (G) Include loads, supports, and inertial effects, which apply on the simulation model.
- (H) In the finite element methods, the shape functions are used to interpolate the displacement fields interior to the element. Across the element boundaries, the displacement fields are continuous but not smooth. The strains and stresses, which are calculated by differentiating the displacement fields, become discontinuous across the element boundaries.
- (I) For an element, strain energies calculated using averaged stresses and unaveraged stresses are different. The difference between these two energy values is called the structural error of the element. The finer the mesh, the smaller the structural error. It is used as an indicator for mesh adequacy.
- (J) Study of how the finite element solutions approach theoretical values as the mesh is getting finer. In 2D, quadrilateral elements converge faster than the triangular. In 3D, the order of convergence speeds are, from faster to slower, hexahedral, prism, pyramid, and tetrahedral.
- (K) A stress that has infinitely large value. They are often found at concave fillets of zero radius, or at points subject to concentrated forces or displacement constraints.

3.6.2 Additional Workbench Exercises

Stress in a Long Cylinder

Consider the problem described in VM25 of the APDL verification manual^[Ref 1]. Find the radial stress and the hoop stress at the inner and outer surfaces and at the middle wall thickness, for both loading cases. Model the problem as (a) an axisymmetric problem and (b) a plane strain problem.

Reference

1. All Help>Verification Manuals>ANSYS Mechanical APDL Verification Manual//I.Verification Test Case Description//I. Introduction//VM25

Chapter 4

3D Solid Modeling

Creating a 3D geometric model is usually much more elaborate than creating a 2D model. Most of the techniques you've learned in Chapters 2 and 3 for 2D cases can be used in 3D cases. Three types of 3D bodies supported by Workbench are *solid bodies*, *surface bodies*, and *line bodies*; they may coexist in a 3D model. In this chapter, we will focus on models consisting of solid bodies, except Section 4.3, in which a surface body and a solid body constitute a 3D model. Chapters 6 and 7 will discuss surface bodies and line bodies, respectively.

Purpose of This Chapter

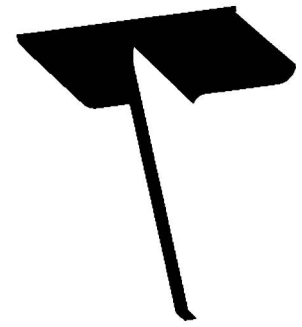
This chapter guides students to familiarize themselves with 3D solid modeling using DesignModeler. Four mechanical parts are created in this chapter. These models will be used for simulations in the next and later chapters.

About Each Section

We start with a beam bracket model in Section 4.1 to introduce the basics of 3D bodies creation. Section 4.2 creates a more elaborated model, the cover of a pressure cylinder, to obtain some degree of proficiency at the 3D modeling techniques. Section 4.3 creates a model for a lifting fork, a combination of a solid body and a surface body. Section 4.4 overviews 3D solid bodies creation and manipulation tools in a systematic way, intending to cover what was missed in the first three sections. Section 4.5 provides an additional exercise.

Section 4.1

Beam Bracket

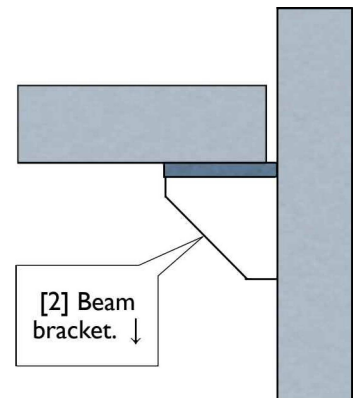


4.1.1 About the Beam Bracket

[1] When a steel beam-column structure, such as a high-rise building or a manufacturing plant, is constructed, its columns are erected before the beams can be elevated, positioned, and welded. The function of a beam bracket [2] is to precisely position a beam and safely transfer the loads from the beam to the column. The loads are determined by a thorough analysis of the entire structure subject to design loads, such as dead load, live load, earthquake, wind load, etc.

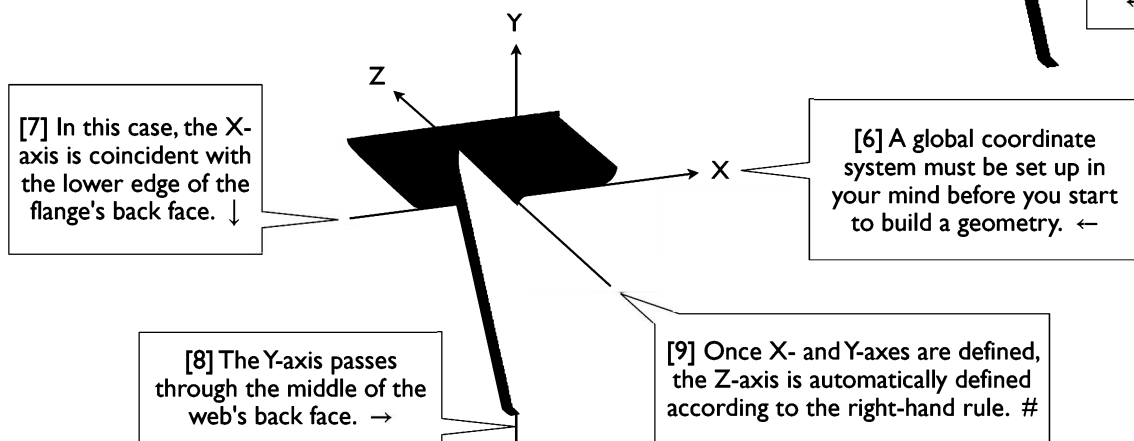
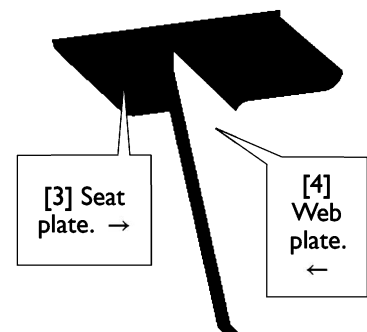
The beam bracket consists of a seat plate (the flange) [3] and a web plate [4]. The design considerations include: (a) Would the maximum stress exceed the allowable stress? (b) Would the web buckle under the load?

In this section, we will create a 3D solid model for the beam bracket. The model will be used for a static structural analysis in Section 5.1 and a buckling analysis in Section 10.3. The 3D solid model will be simplified to a surface model in Section 6.2 and the simulation results will be compared with those in Section 5.1. →

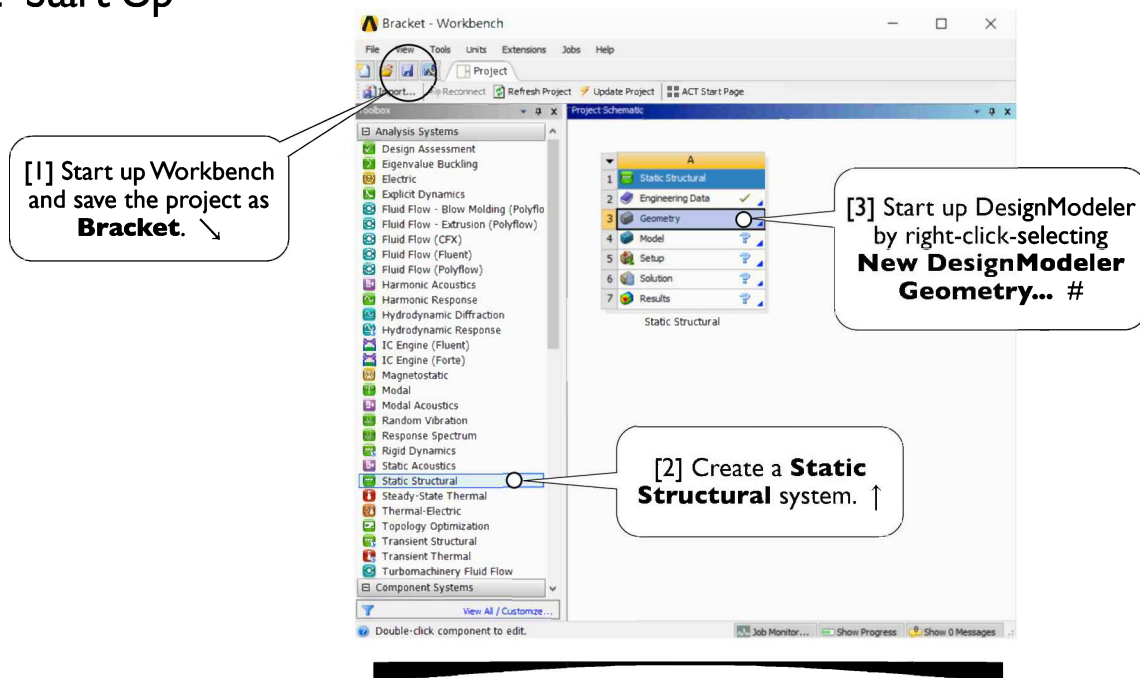


Global Coordinate System

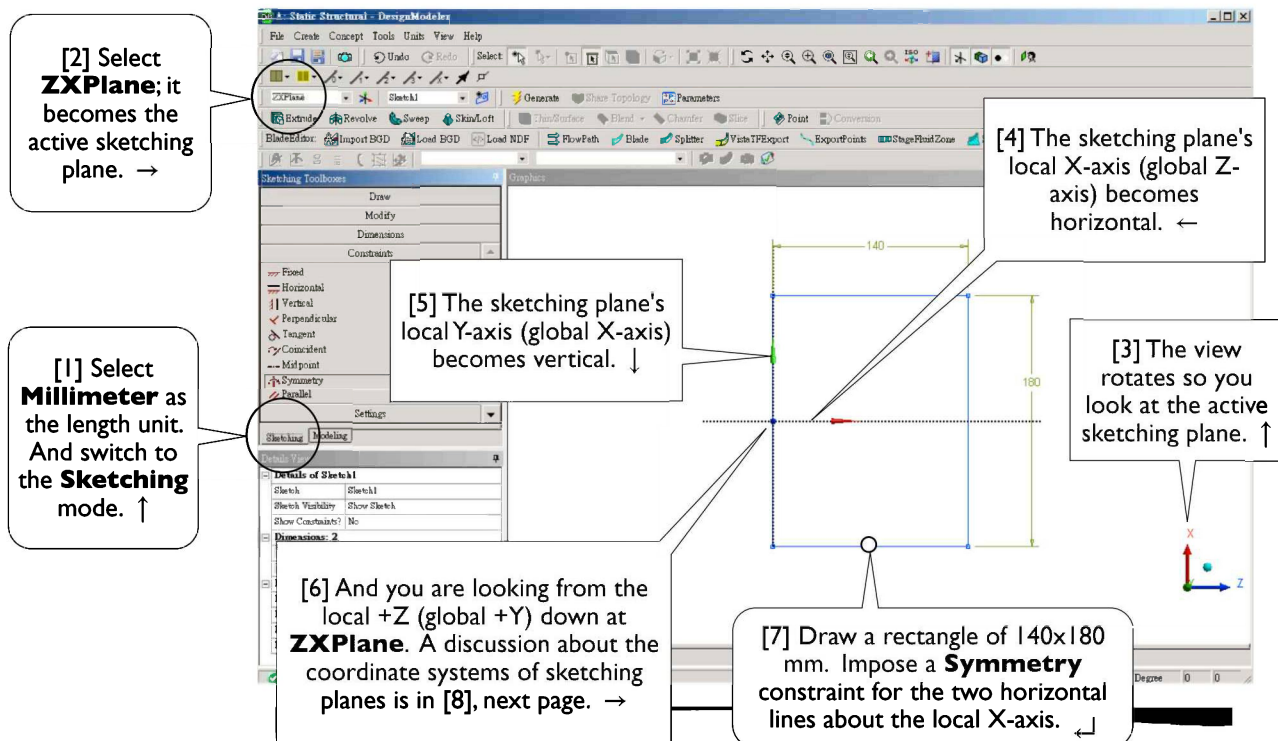
[5] To create a geometric model, you need a global coordinate system [6]. For 2D, it is so trivial that we didn't even mention it. For 3D, you need to pay more attention to the global coordinate system. Many students are easily disoriented when working with 3D models on a 2D computer screen. In this section, the global coordinate system is set up as shown in [7-9]. ↘



4.1.2 Start Up



4.1.3 Sketch the Seat Plate on ZXPlane

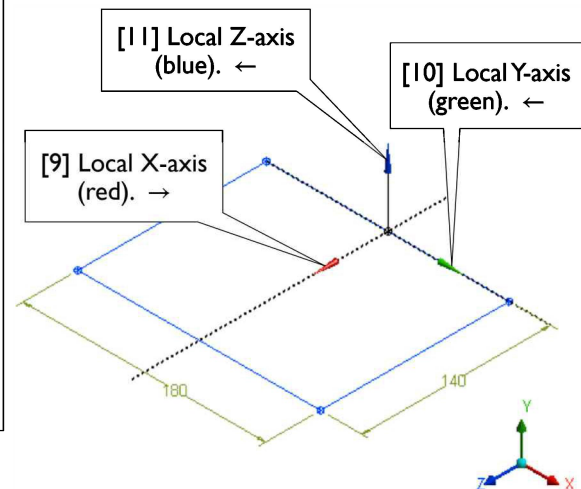


The Local Coordinate System of a Sketching Plane

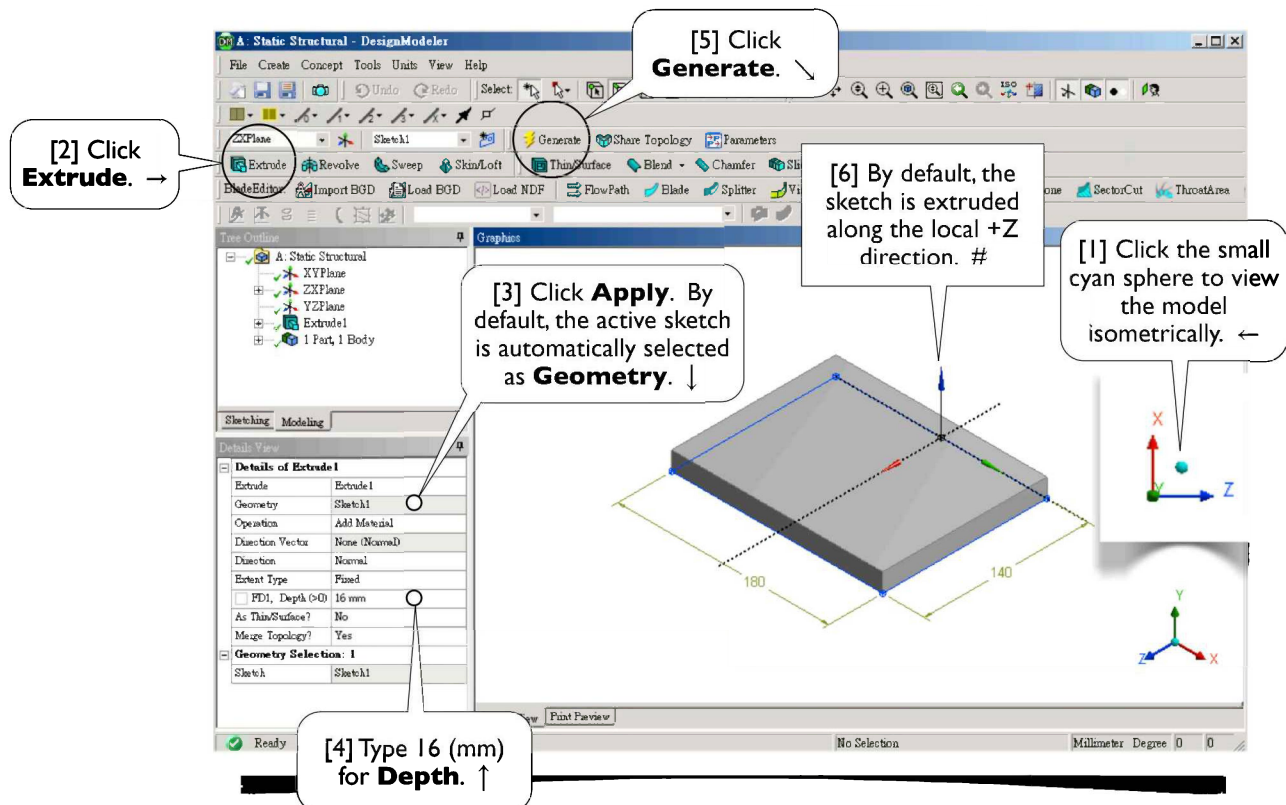
[8] Each sketching plane has a local coordinate system (3.1.1.1 [13], page 119). For the three pre-defined planes (**XYPlane**, **YZPlane**, and **ZXPlane**), their first two letters refer to two global axes, which become the local X-axis and Y-axis, respectively. The local Z-axis is then defined according to the right-hand rule. For example, **ZXPlane**'s local X-axis and local Y-axis coincide with the global Z-axis and global X-axis, respectively [9-11]. ↓

Plane View Sketching vs. 3D View Sketching

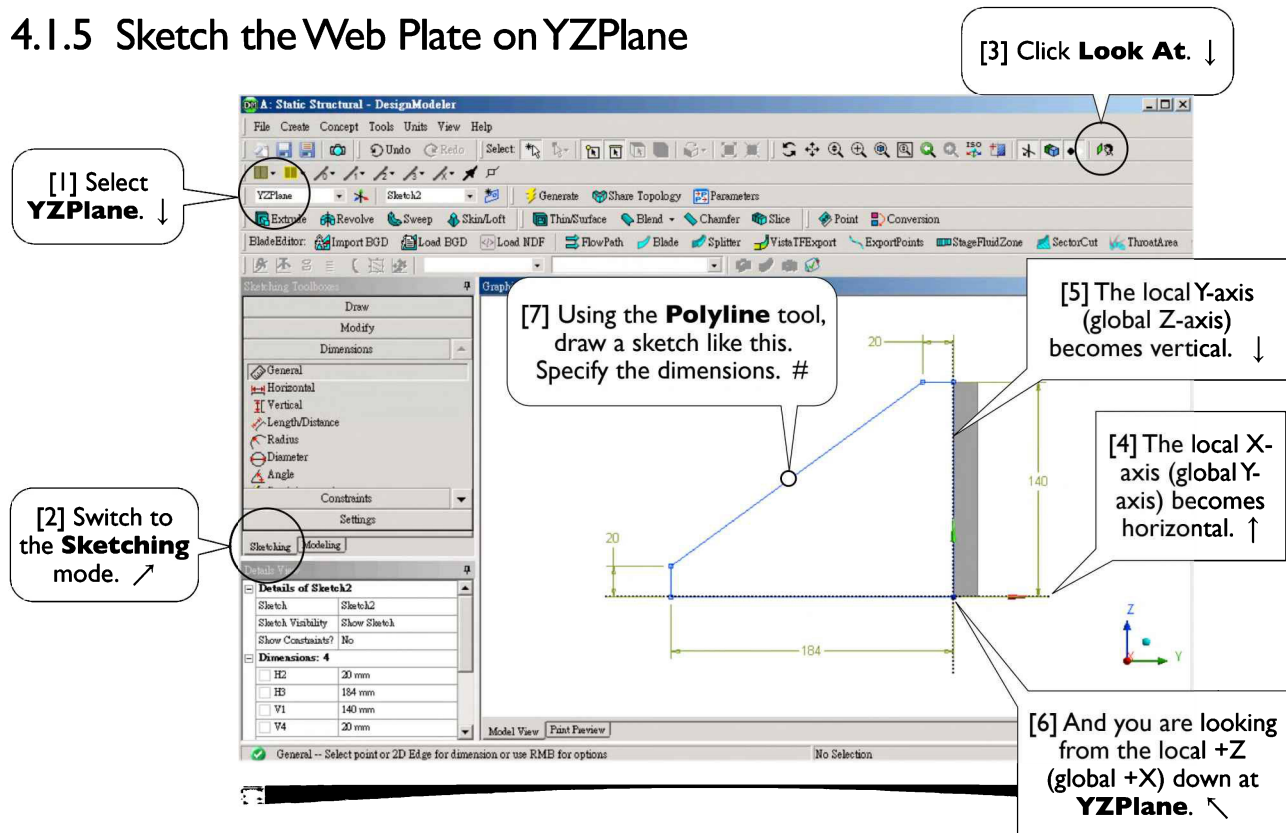
[12] When the view rotates to look at the sketching plane [3], the local X-axis and local Y-axis of the sketching plane becomes horizontal and vertical, respectively [4-5], and the local Z-axis points out toward you. It is a simple rule but may take time to get accustomed to. If you have trouble sketching with plane view [4-6], alternatively you may sketch in 3D view, as shown in [9-11]. The problem of sketching in 3D view is that all angles are distorted. For example, a right angle is no longer 90°. For most people, sketching with plane view should be more convenient. However, choose whichever is suitable for you. In this book, we will stick to the plane view sketching. #



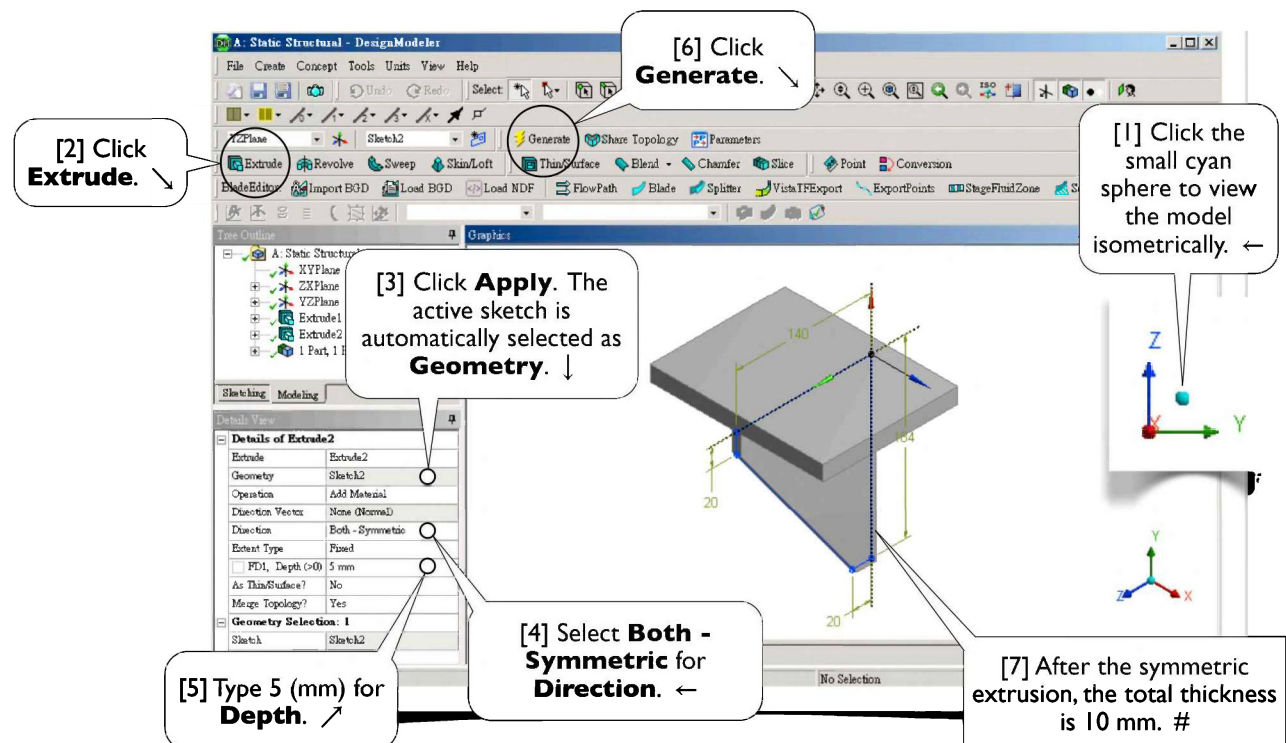
4.1.4 Extrude the Sketch to Create Seat Plate



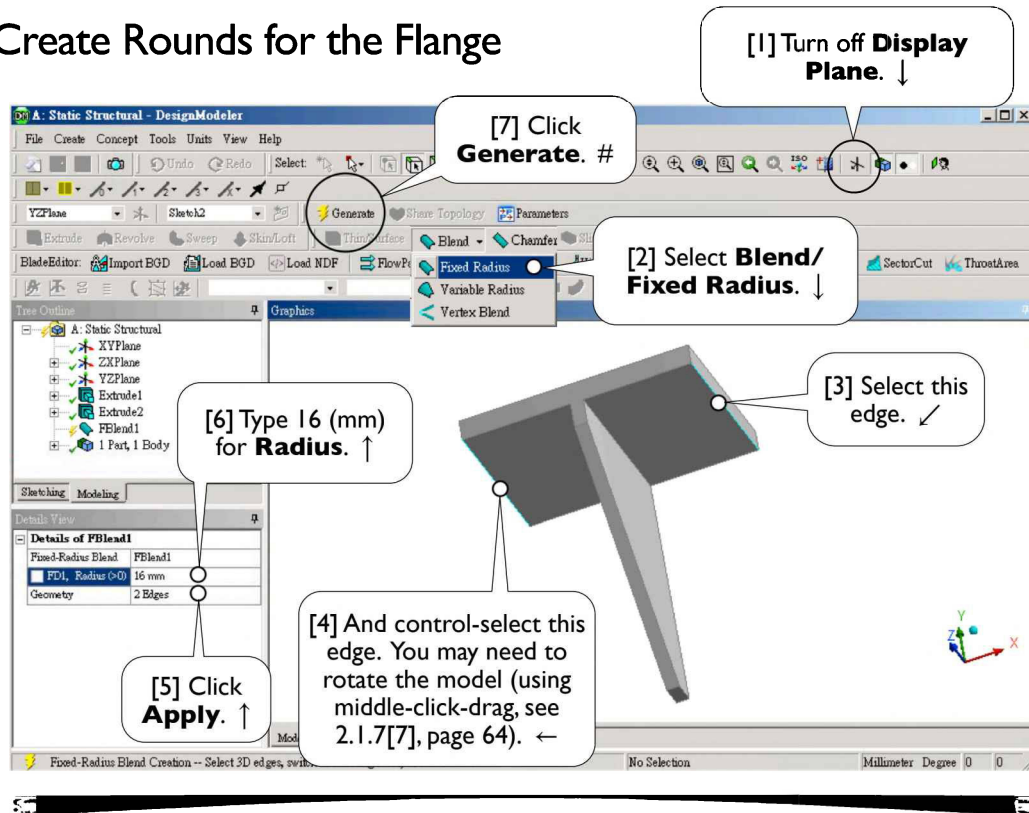
4.1.5 Sketch the Web Plate on YZPlane



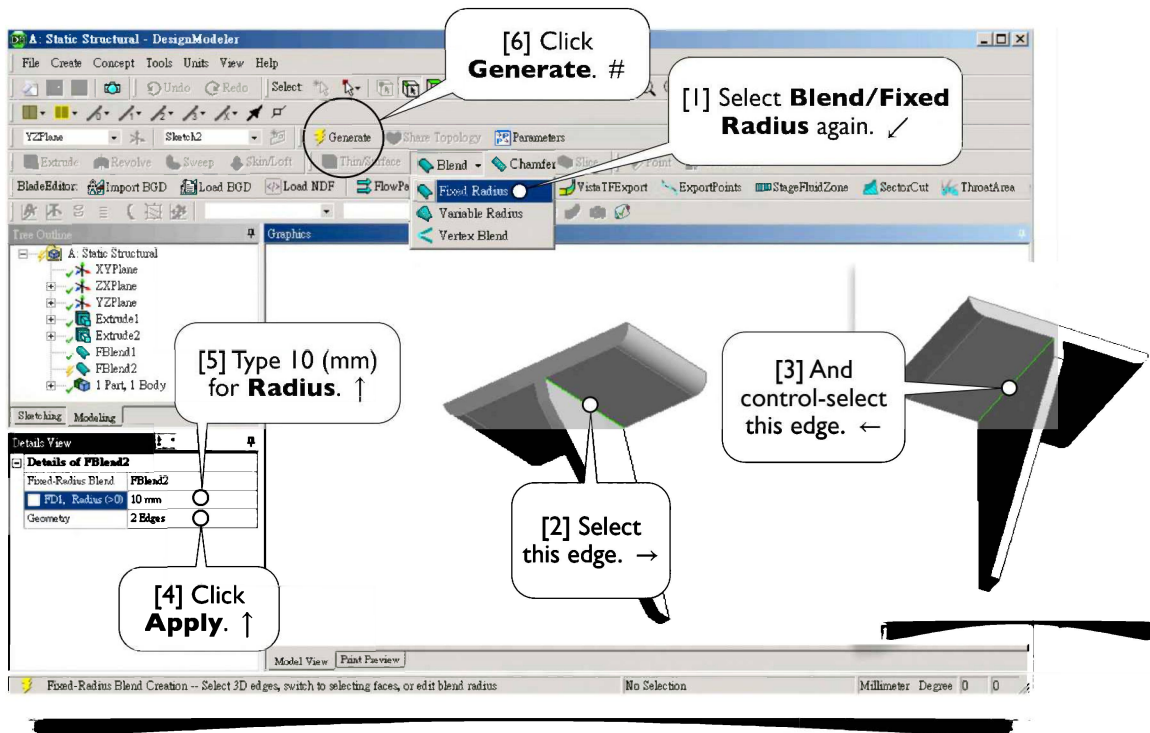
4.1.6 Extrude the Sketch to Create Web Plate



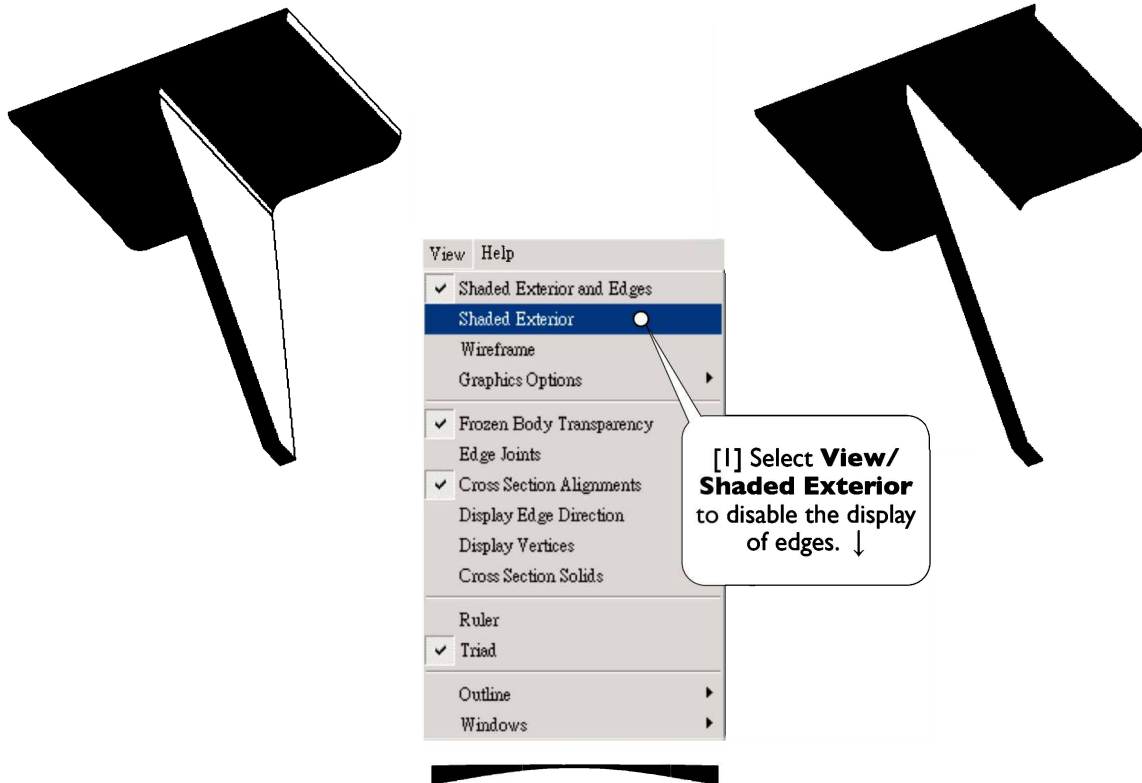
4.1.7 Create Rounds for the Flange



4.1.8 Create Fillets



4.1.9 Turn Off Edges Display

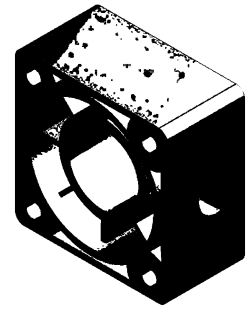


Wrap Up

[2] Close DesignModeler, save the project, and exit Workbench. #

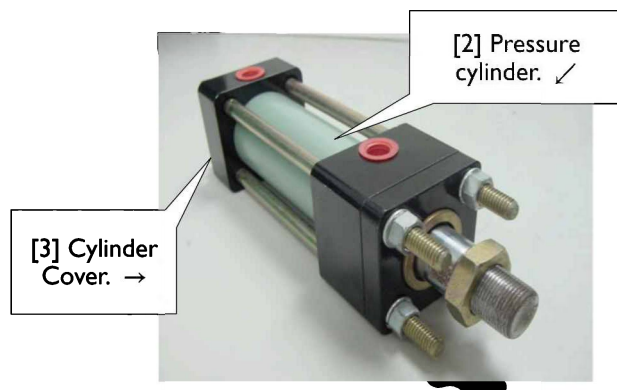
Section 4.2

Cover of Pressure Cylinder

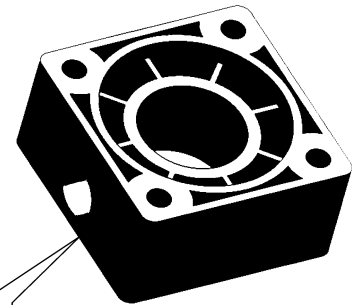


4.2.1 About the Cylinder Cover

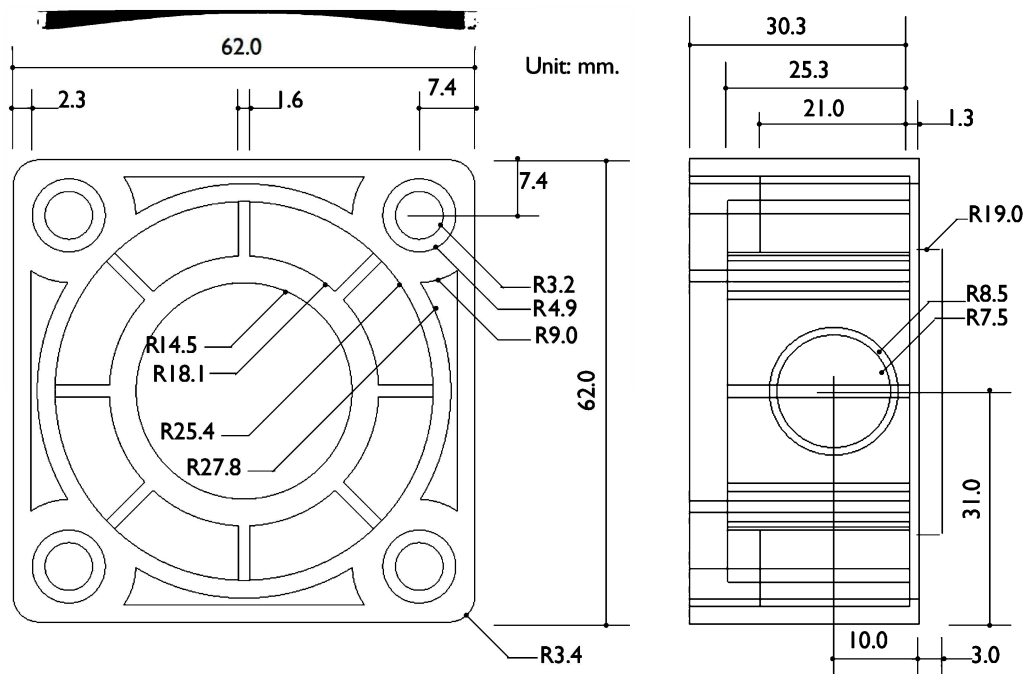
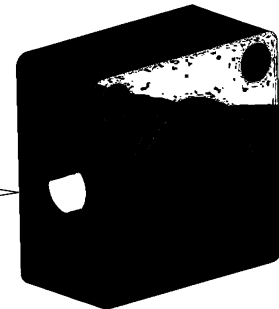
[1] The pressure cylinder [2] contains a gas of 0.5 MPa. The cylinder cover [3-5] is made of a carbon-fiber reinforced plastic. In this section, we want to create a 3D solid model for the cover; the model will be used for a static structural simulation in Section 5.2 to investigate the deformation under pressure. ✓



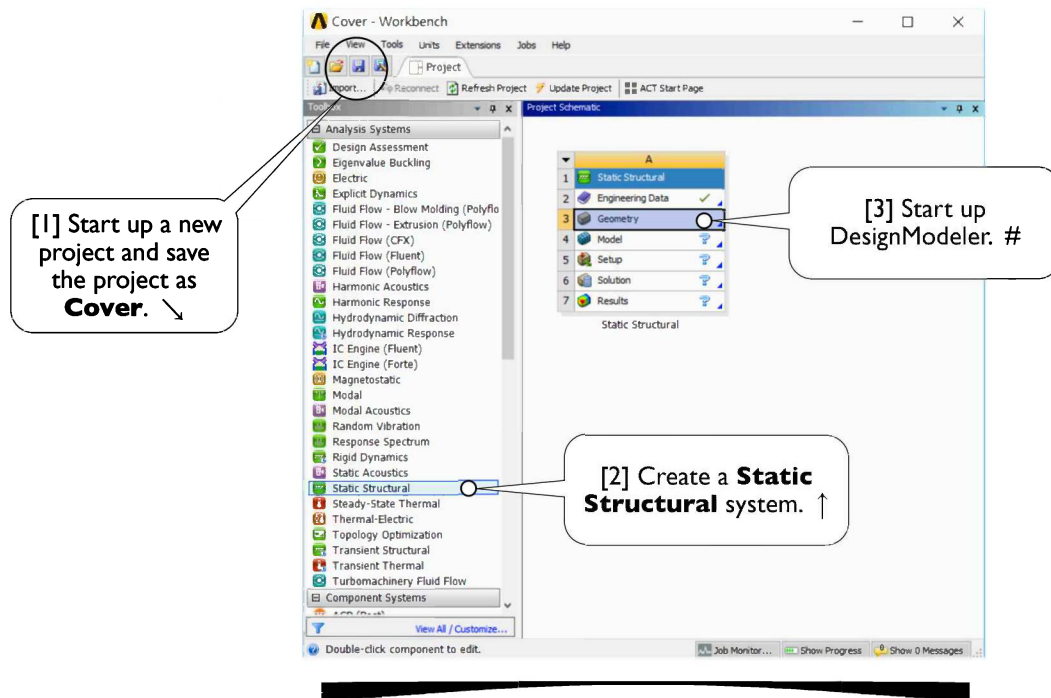
[4] A 3D view of the cover. ↓



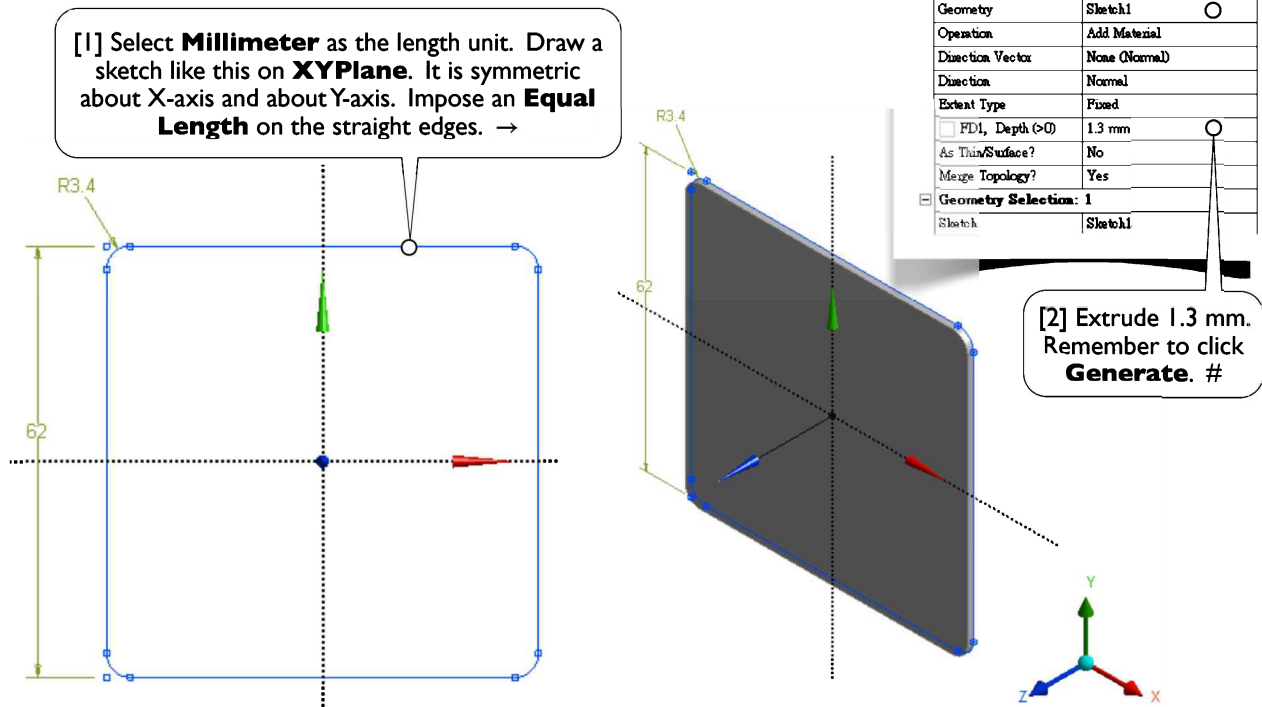
[5] Another view of the cover. #



4.2.2 Start Up



4.2.3 Create Base Plate



4.2.4 Create a New Sketching Plane

[1] Click **New Plane**. **Plane4** is created. ↓

[4] Click this face at a location near the lower-left corner. ↓

[2] Select **From Face**. ↓

[6] And click **Apply**. ↓

[3] Click the yellow area to bring up the **Apply/Cancel** buttons. ↑

[5] The location where you click determines not only the origin but also the axes. If your coordinate system is not like this, click again until the coordinate system is exactly the same as here. Note that the red-arrow (local X-axis) points downward. ←

[10] Also note that **Outline Plane** is the default **Subtype**; see an explanation in 4.2.5[5], next page. #

[7] Set up the transformations like this. (After setting up a transformation, another will show up.) The purpose of the transformations is to set up a coordinate system as shown in [8]. ✓

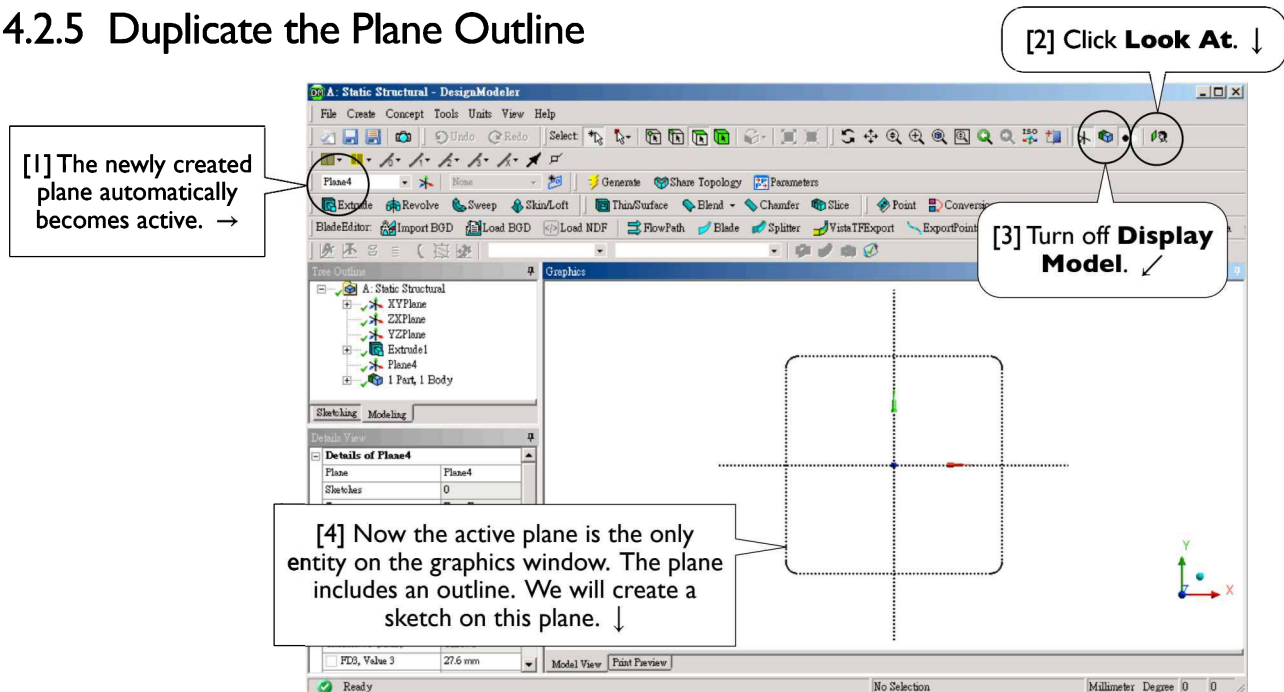
[8] You can visually see the change of the local coordinate system when you type a transformation in [7]. Note that this local coordinate system is the same as the global except their origins are 1.3 mm apart in Z-direction. ✓

[9] Click **Generate** to complete the creation of the plane. ↑

Details of Plane4	
Plane	Plane4
Type	From Face
Subtype	Outline Plane
Base Face	Selected
Use Axis Centers for Origin?	Yes
Transform 1 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

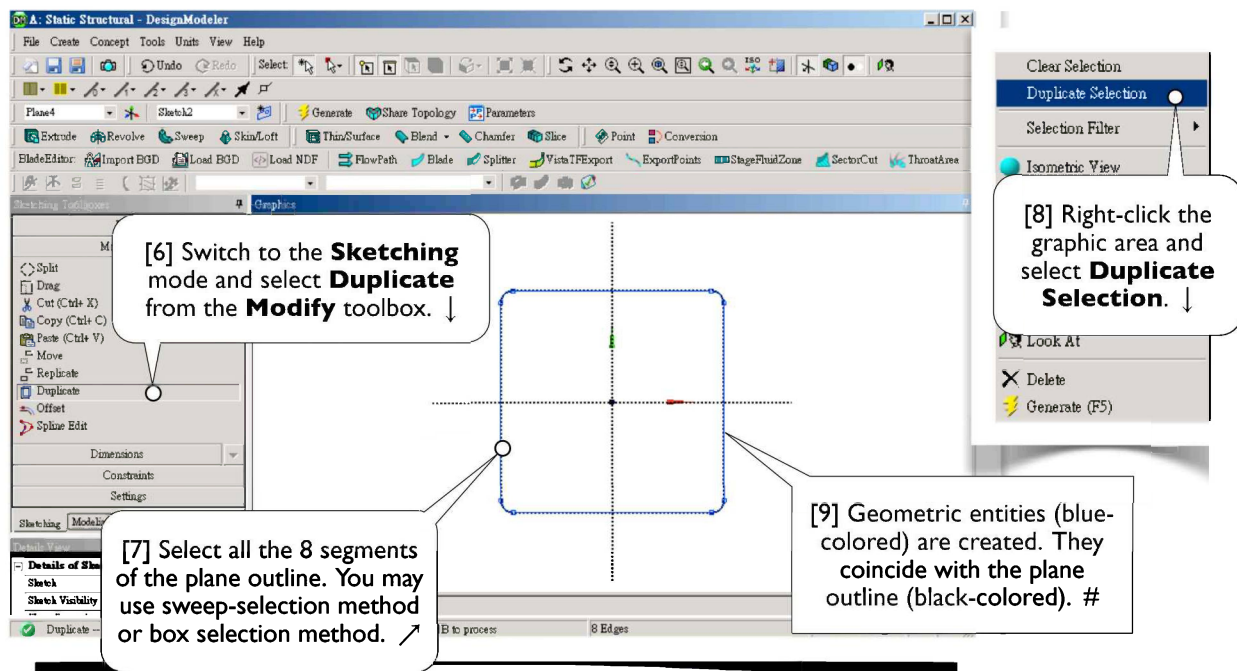
Details of Plane4	
Plane	Plane4
Sketches	0
Type	From Face
Subtype	Outline Plane
Base Face	Selected
Use Axis Centers for Origin?	Yes
Transform 1 (RMB)	Rotate about Z
FD1, Value 1	90 °
Transform 2 (RMB)	Offset X
FD2, Value 2	31 mm
Transform 3 (RMB)	Offset Y
FD3, Value 3	27.6 mm
Transform 4 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

4.2.5 Duplicate the Plane Outline

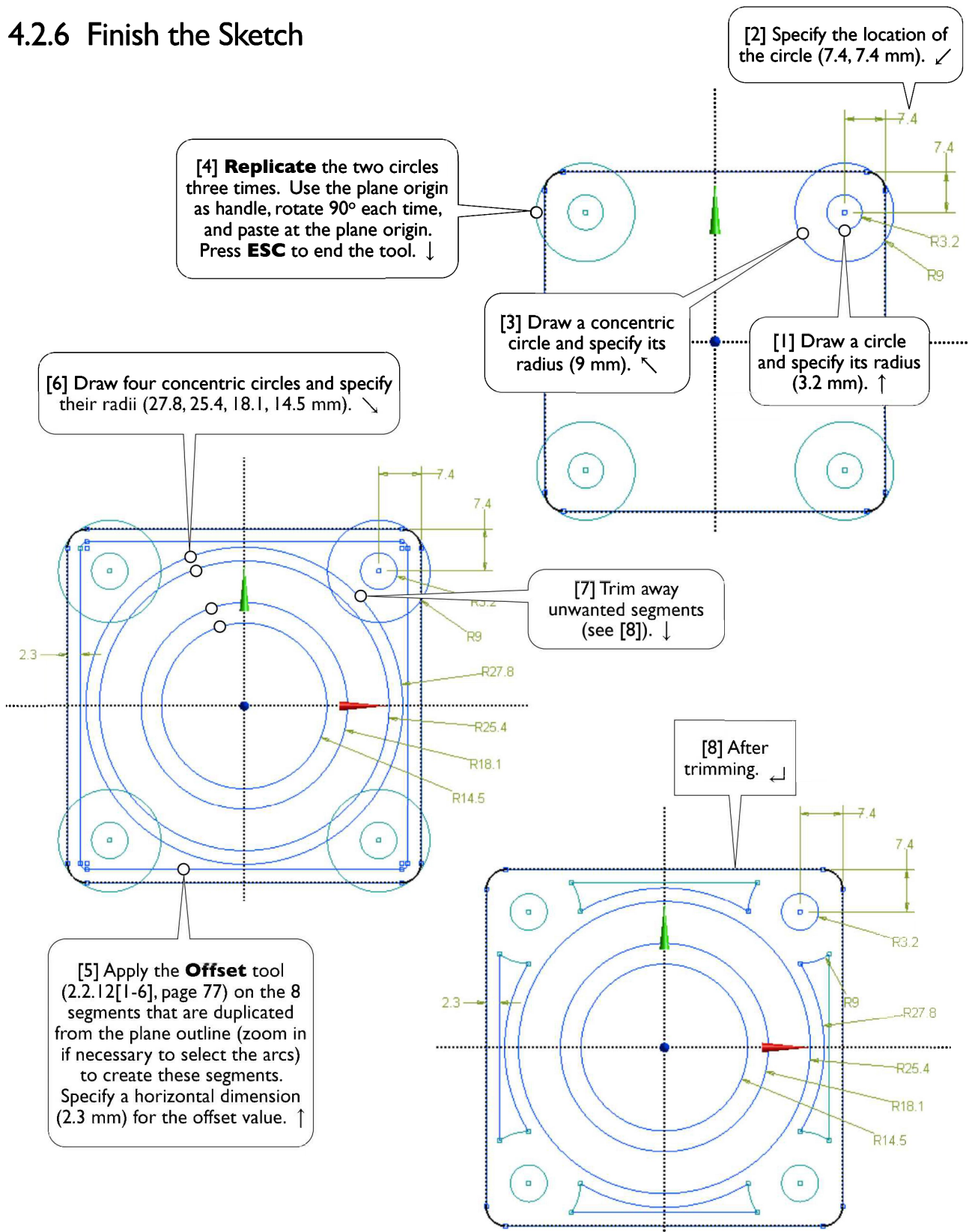


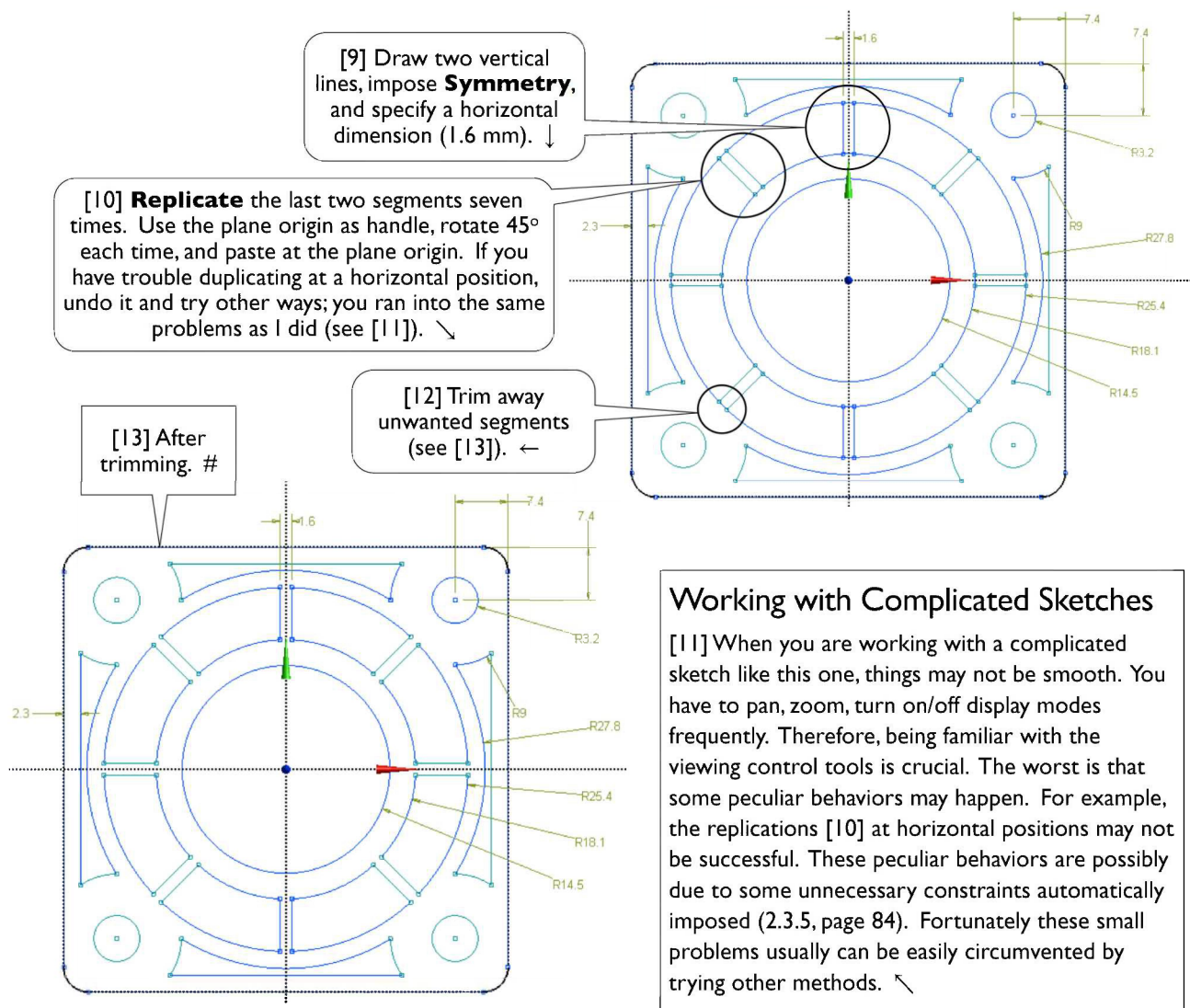
Outline Plane

[5] When a plane is derived from a face and its **Subtype** is **Outline Plane**, the face outline will be included in the plane. The outline is not a geometric entity; it can be used as a reference, or datum, just like an X-axis or Y-axis can be used as a datum. It also can be duplicated to create geometric entities, as demonstrated in [6-9]. ✓



4.2.6 Finish the Sketch





4.2.7 Extrude the Sketch

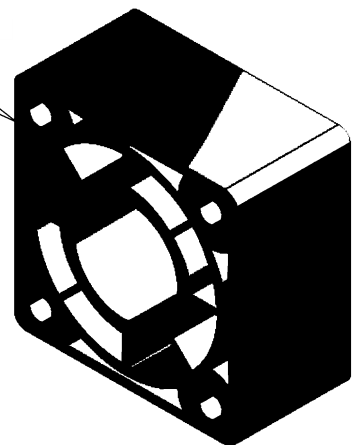
Details of Extrude2	
Extrude	Extrude2
Geometry	Sketch2 <input type="radio"/>
Operation	Add Material
Direction Vector	None (Normal)
Direction	Normal
Extent Type	Fixed
FD1, Depth (>0)	30.3 mm <input type="radio"/>
As Thin/Surface?	No
Merge Topology?	Yes
Geometry Selection: 1	
Sketch	Sketch2

[1] Extrude 30.3 mm. ↘

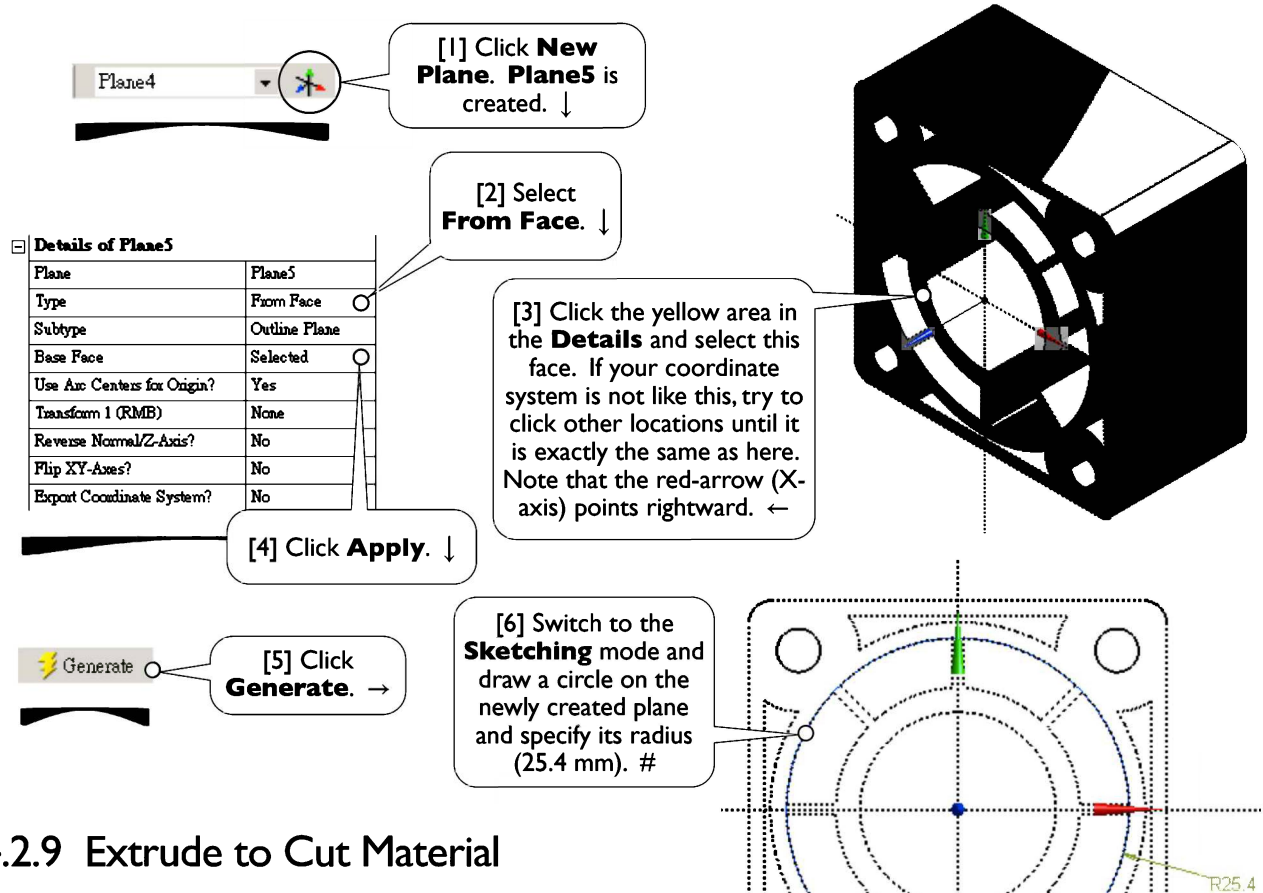
[3] The extruded solid body. #

[2] Click **Generate**. ↑

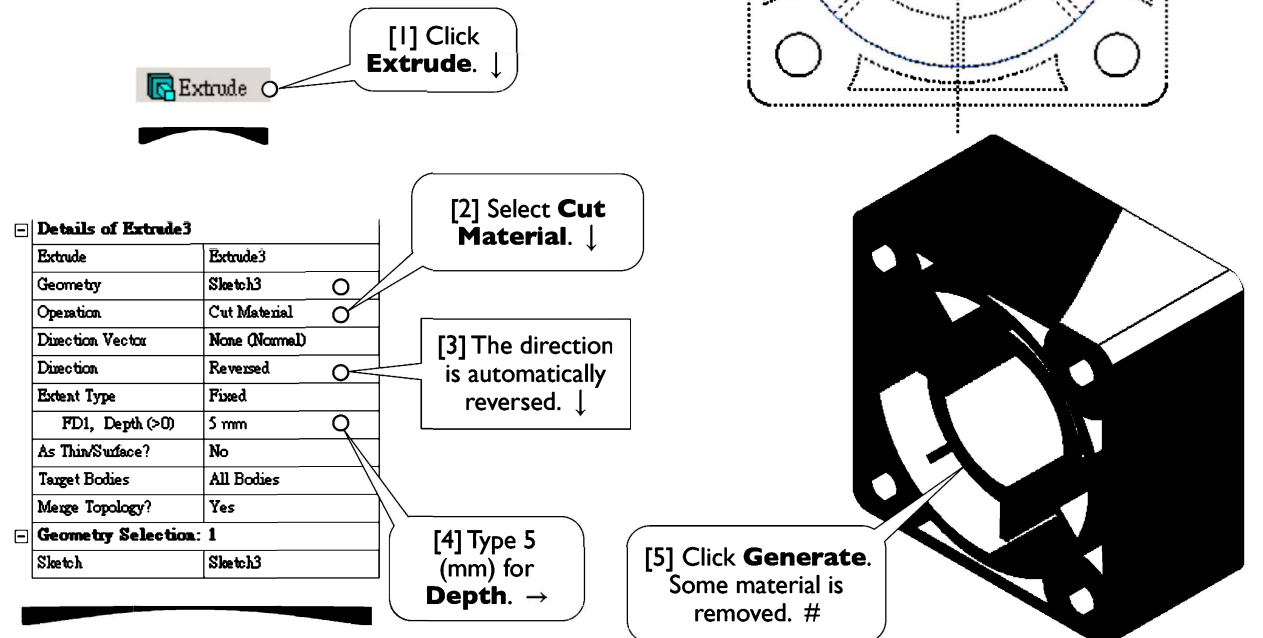
Generate



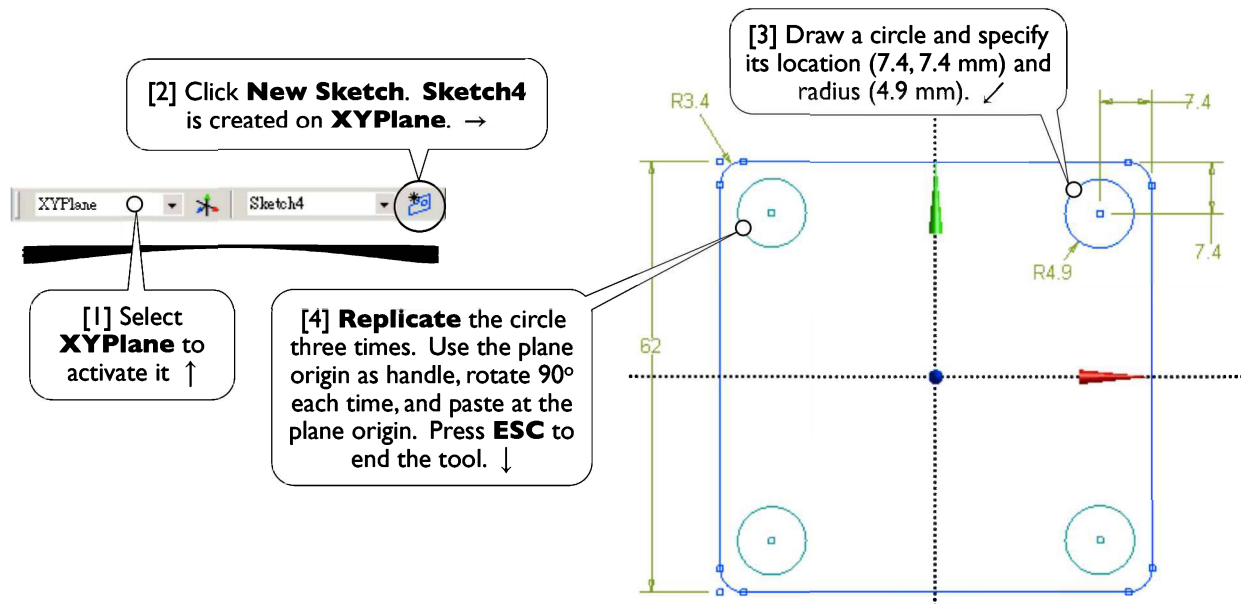
4.2.8 Create a New Plane and Draw a Circle on the Plane



4.2.9 Extrude to Cut Material



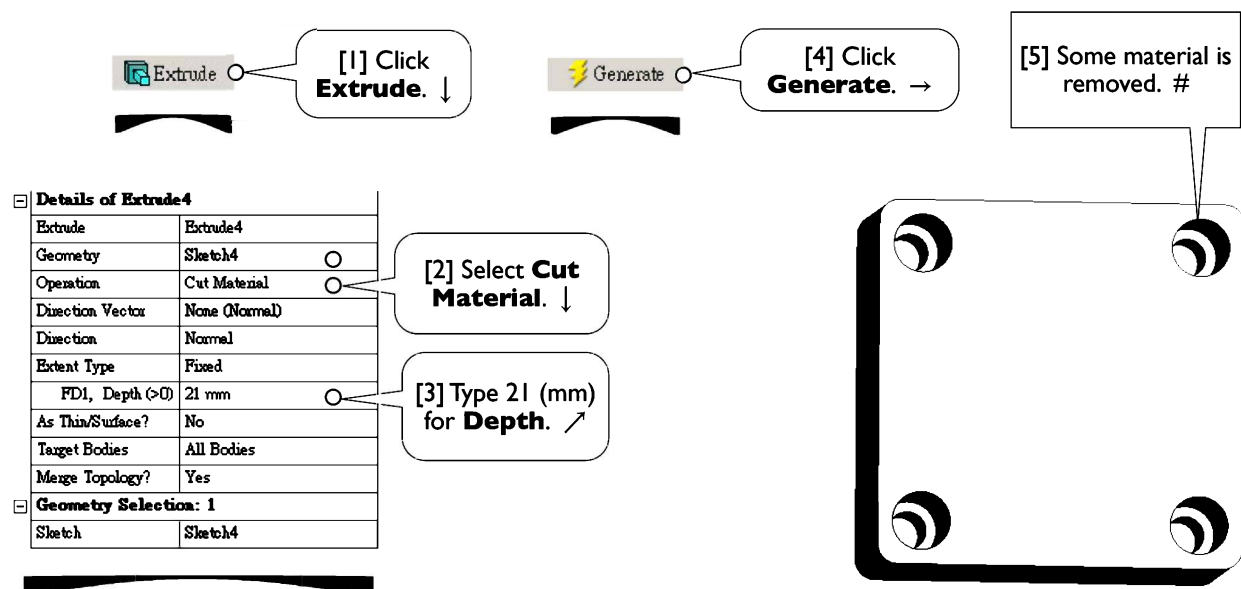
4.2.10 Create New Sketch on XYPlane and Draw Circles



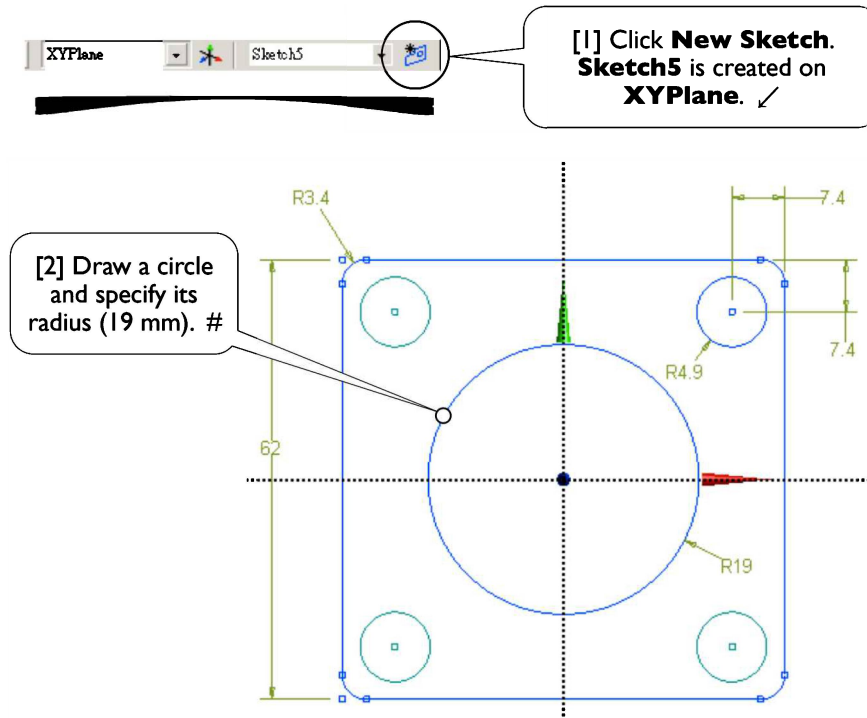
Create Second Sketch on a Plane

[5] A plane may contain multiple sketches. When you begin to draw on a fresh plane, a sketch is automatically created; you don't have to explicitly click **New Sketch** as in [2], although there is no harm in doing that. When you want to add a second sketch on a plane, you **MUST** explicitly click **New Sketch**, otherwise the entities you draw will become part of the first sketch. #

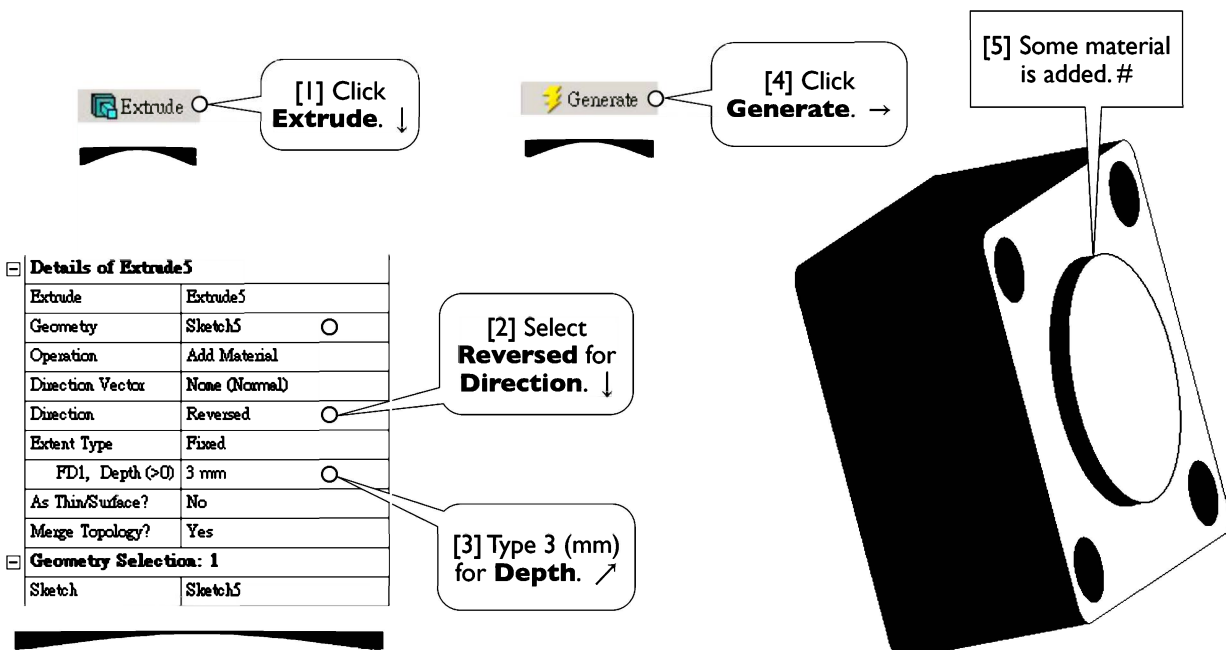
4.2.11 Extrude to Cut Material



4.2.12 Create Another Sketch on XYPlane and Draw a Circle



4.2.13 Extrude to Add Material



4.2.14 Create a Plane on a Side Wall and Two Sketches on the Plane

[1] Click **New Plane**. **Plane6** is created. ↓

[2] Select **From Face**. →

[3] Click the yellow area in the **Details** and click this face near here. If your coordinate system is not like this, try to click different locations until it is exactly the same as here. Note that the red arrow (X-axis) points downward. ←

[4] Click **Apply**. ↘

Details of Plane6	
Plane	Plane6
Type	From Face
Subtype	Outline Plane
Base Face	Selected
Use Arc Centers for Origin?	Yes
Transform 1 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

[6] Draw a circle on the newly created plane. Specify its position (27.6, 20.3 mm) and radius (8.5 mm). This sketch is **Sketch6**. ✓

[5] Click **Generate**. ←

[7] Click **New Sketch**. **Sketch7** is created on the same plane. →

[8] Draw a smaller concentric circle and specify its radius (7.5 mm). #

4.2.15 Complete the Model

[1] Click **Extrude**. ↓

[2] Select **Sketch6** from the model tree (the larger circle) and click **Apply**. ↓

[3] Select **Reversed**. ↓

[4] Type 15 (mm) for **Depth**. ↓

[6] A solid cylinder is added to the model. ✓

Details of Extrude6	
Extrude	Extrude6
Geometry	Sketch6
Operation	Add Material
Direction Vector	None (Normal)
Direction	Reversed
Extent Type	Fixed
FD1, Depth (>0)	15 mm
As Thin/Surface?	No
Merge Topology?	Yes
Geometry Selection: 1	
Sketch	Sketch6

[5] Click **Generate**. ✓

[7] Click **Extrude** again. →

[12] Click **Generate**. ↓

[8] Select **Sketch7** from the model tree and click **Apply**. ↓

[9] Select **Cut Material**. ↓

[10] **Reversed** is automatically selected. ↓

Details of Extrude7	
Extrude	Extrude7
Geometry	Sketch7
Operation	Cut Material
Direction Vector	None (Normal)
Direction	Reversed
Extent Type	Fixed
FD1, Depth (>0)	31 mm
As Thin/Surface?	No
Target Bodies	All Bodies
Merge Topology?	Yes
Geometry Selection: 1	
Sketch	Sketch7

[11] Type a **Depth** cutting the material through the length of the solid cylinder [6], say 31 (mm). ←

[13] The solid cylinder becomes hollow. ↓

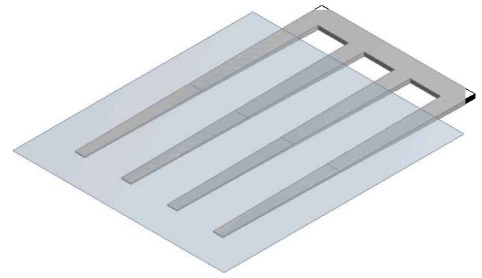
[14] Select **View/Shaded Exterior**. ↓

Wrap Up

[15] Close DesignModeler, save the project, and exit Workbench. #

Section 4.3

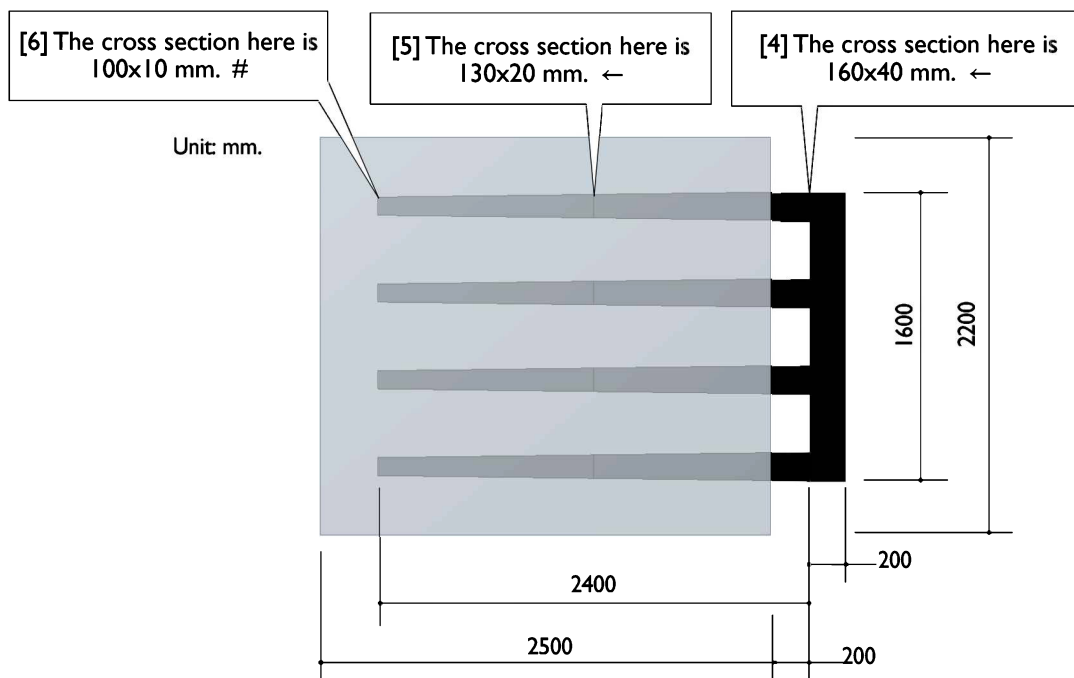
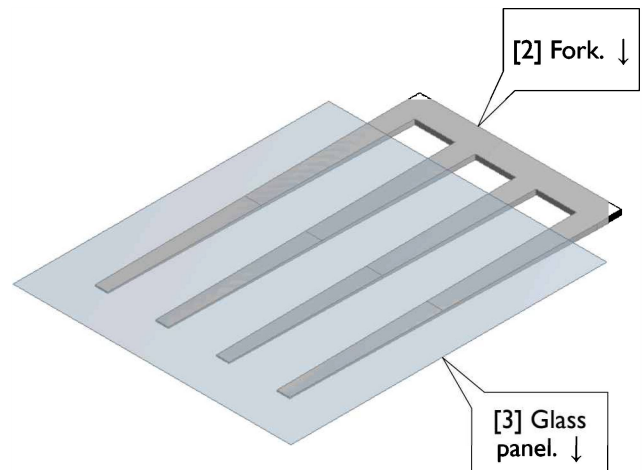
Lifting Fork



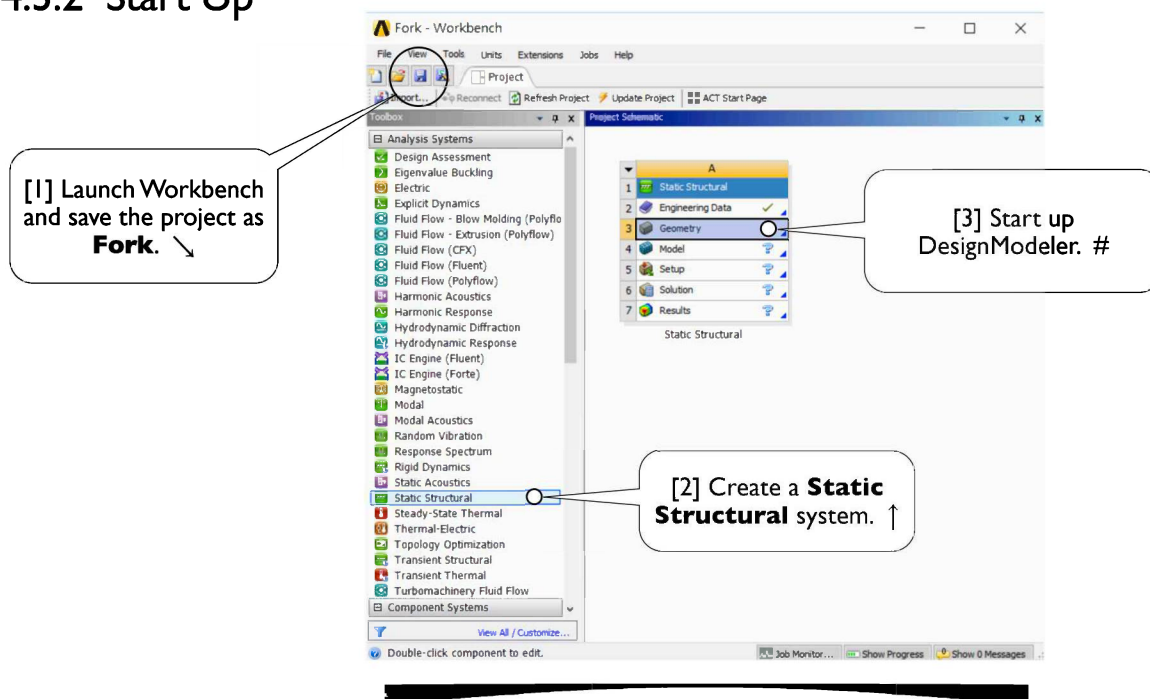
4.3.1 About the Lifting Fork

[1] The lifting fork [2-6] is used in an LCD (liquid crystal display) manufacturing factory to handle glass panels. The glass panel is large (2.5 m x 2.2 m) and thin (1.0 mm) and the engineers are concerned about its deflections during the dynamic handling, which is a critical parameter in the design of a precision machine handling process. Another important parameter is the time before the panel's vibrations cease.

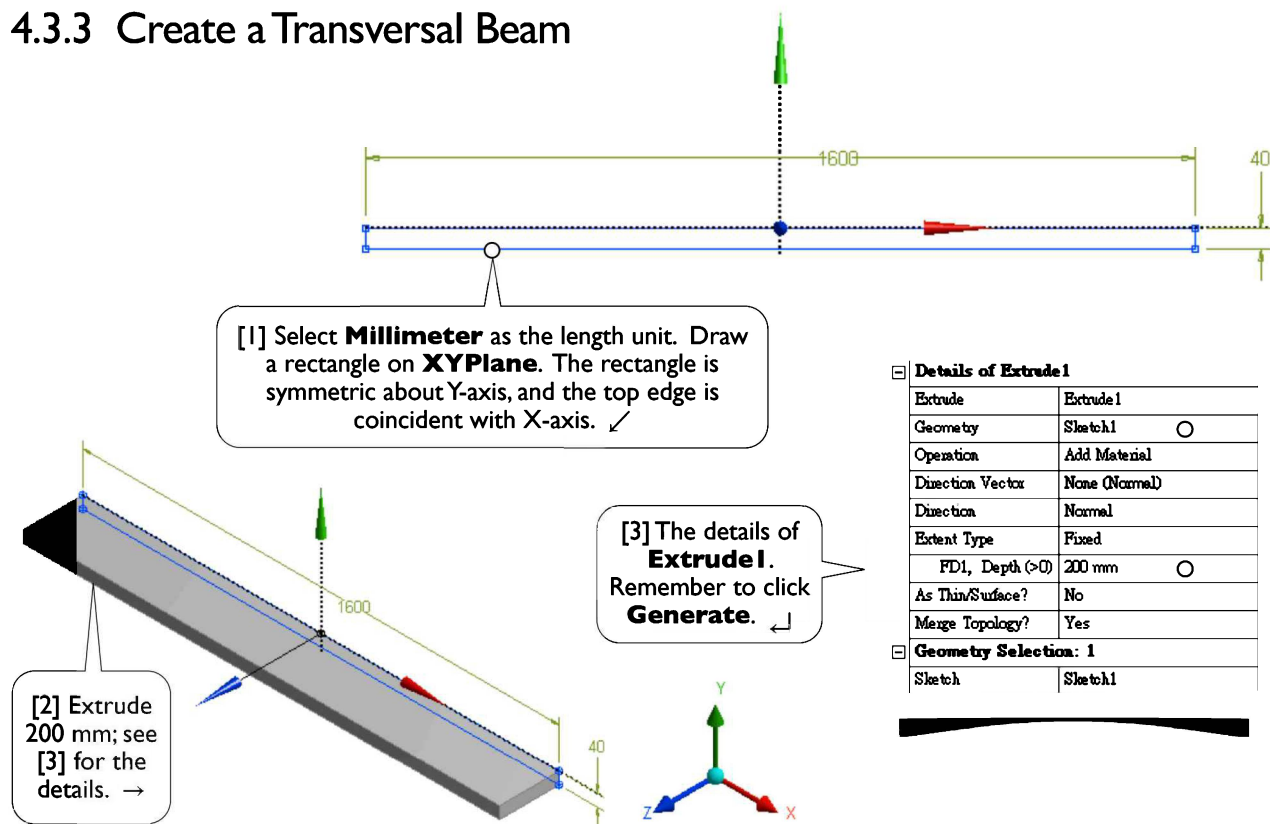
In this section, we will model the fork as a solid body and the glass panel as a surface body. The model will be used for simulations in Section 12.2. →



4.3.2 Start Up



4.3.3 Create a Transversal Beam



Skin/Loft

[4] Now we want to create a prong (or finger). The prong is then duplicated to create 3 additional prongs. The prong's cross section is not uniform, and it cannot be created using **Extrude** or **Sweep**. A way to create a solid body of different cross sections along its length is using **Skin/Loft**, which takes a series of profiles from different planes and creates a solid (or surface) that fits through these profiles.

You may view **Sweep** as a special case of **Skin/Loft** and **Extrude** as a special case of **Sweep**. #

4.3.4 Create Three Planes Based on a Face of the Beam

[1] All three planes will be created based on this face. When you select this face, make sure the origin of the coordinate system is at the bottom-right corner and the three axes are, respectively, parallel to the global axes. ↘

[2] Create **Plane4**; see [3]. ↓

[4] Create **Plane5**; see [5]. ↓

[6] Create **Plane6**; see [7]. ↓

[7] Details of **Plane6**. #

Details of Plane6	
Plane	Plane6
Sketches	0
Type	From Face <input type="radio"/>
Subtype	Outline Plane
Base Face	Selected <input type="radio"/>
Use Axis Centers for Origin?	Yes
Transform 1 (RMB)	Offset Z <input type="radio"/>
FD1, Value 1	2400 mm <input type="radio"/>
Transform 2 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

[5] Details of **Plane5**. ↙

Details of Plane5	
Plane	Plane5
Sketches	0
Type	From Face <input type="radio"/>
Subtype	Outline Plane
Base Face	Selected <input type="radio"/>
Use Axis Centers for Origin?	Yes
Transform 1 (RMB)	Offset Z <input type="radio"/>
FD1, Value 1	1200 mm <input type="radio"/>
Transform 2 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

[3] Details of **Plane4**. ↙

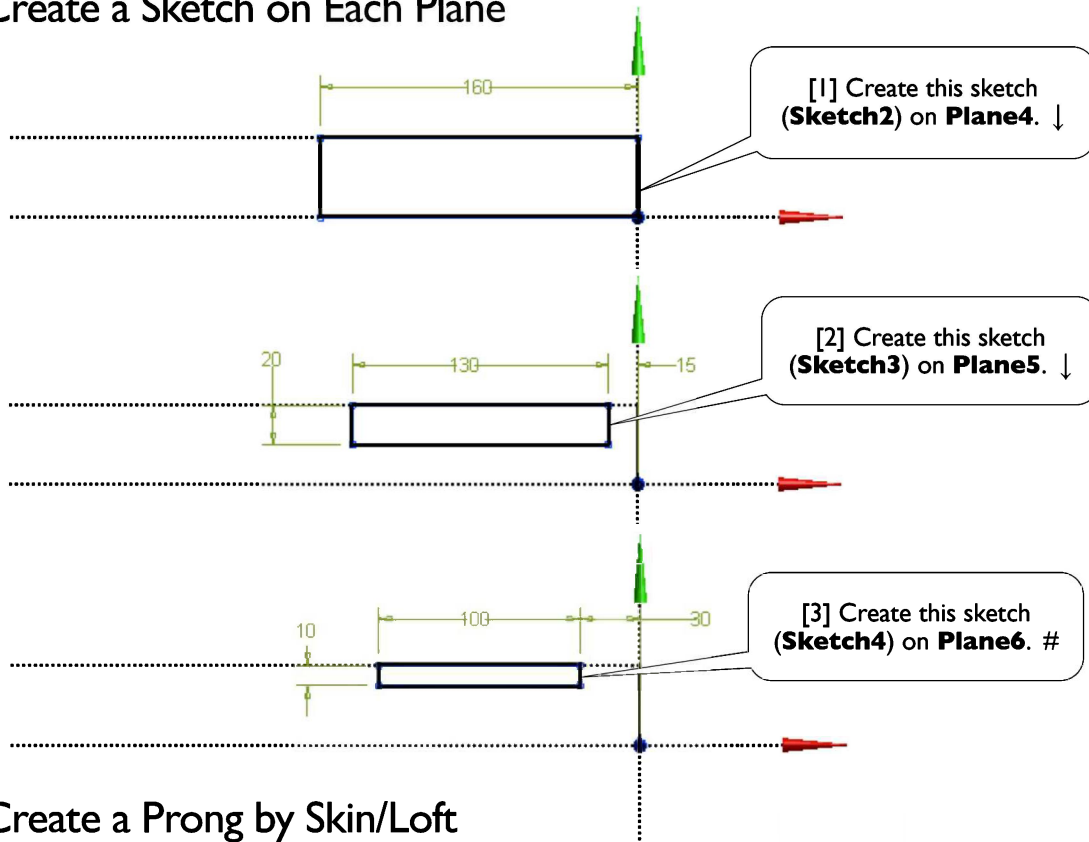
Details of Plane4	
Plane	Plane4
Sketches	0
Type	From Face <input type="radio"/>
Subtype	Outline Plane
Base Face	Selected <input type="radio"/>
Use Axis Centers for Origin?	Yes
Transform 1 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

Generate

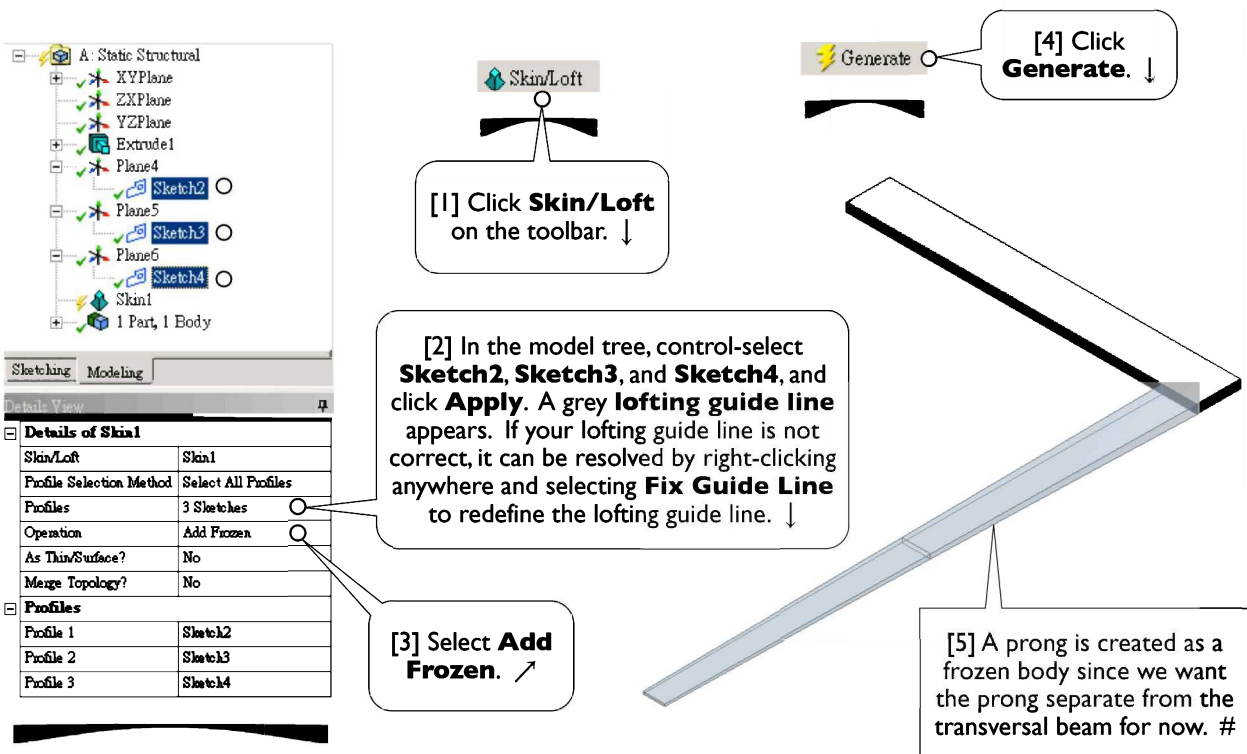
Generate

Generate

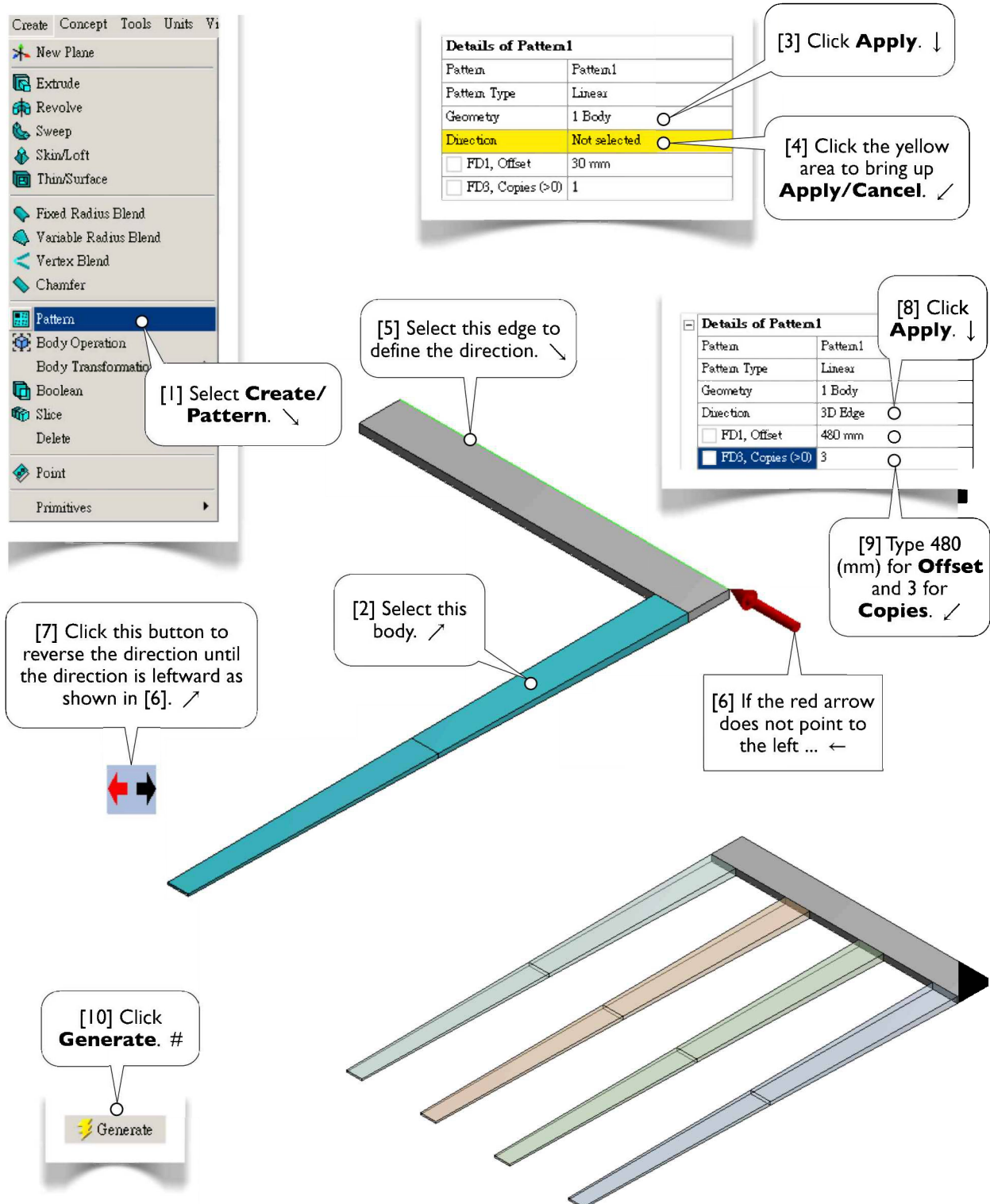
4.3.5 Create a Sketch on Each Plane



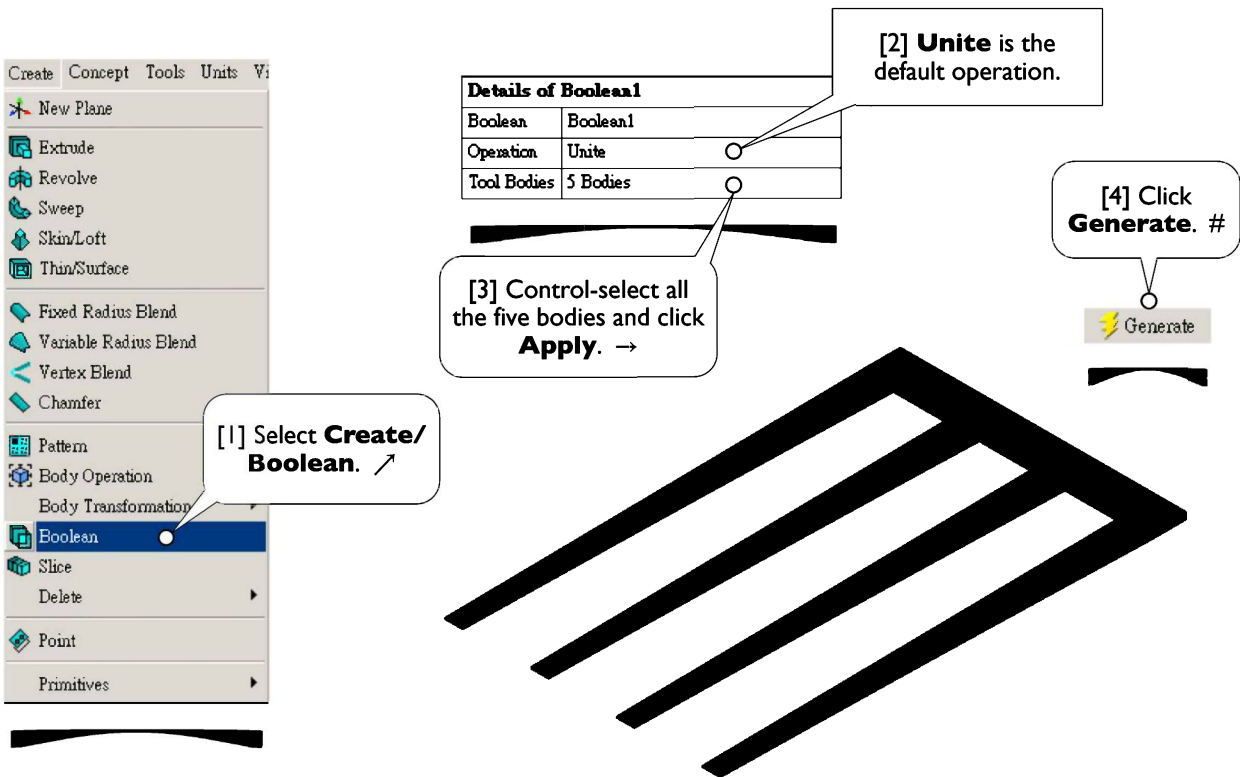
4.3.6 Create a Prong by Skin/Loft



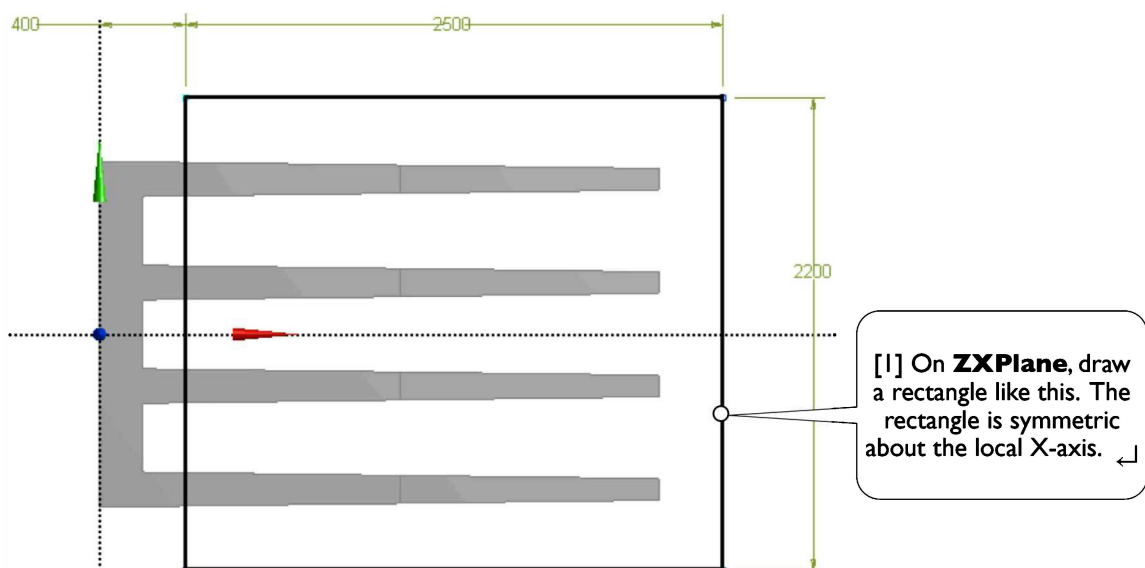
4.3.7 Duplicate the Prong Using Pattern



4.3.8 Combine the Bodies Using **Boolean**



4.3.9 Create a Surface Body for the Glass Panel



[2] Select **Concept/Surfaces From Sketches**.

[3] Select the newly created sketch (**Sketch5**) from the model tree and click **Apply**.

Details of SurfaceSk1	
Surface From Sketches	SurfaceSk1
Base Objects	1 Sketch
Operation	Add Material
Orient With Plane Normal?	Yes
Thickness (>=0)	1 mm

[4] Leave **Add Material** unchanged (see [7]).

[5] Type 1 (mm) for the thickness of the glass panel. This thickness will be exported to **Mechanical**.

[6] Click **Generate**.

[7] Although we set **Add Material** for **Operation** [4], the surface body does not add to the solid body. It is in effect an **Add Frozen** operation and the glass is created as a separate body. This is because different body types cannot be combined to form a single body.

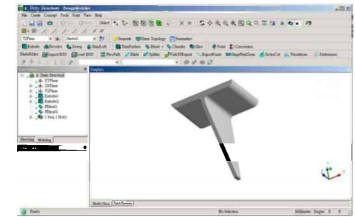
[8] Right-click to rename the two bodies as **Fork** and **Glass**, respectively. Note that the two bodies are treated as two parts. Each part will be meshed independently.

Wrap Up

[9] Close DesignModeler, save the project, and exit Workbench. #

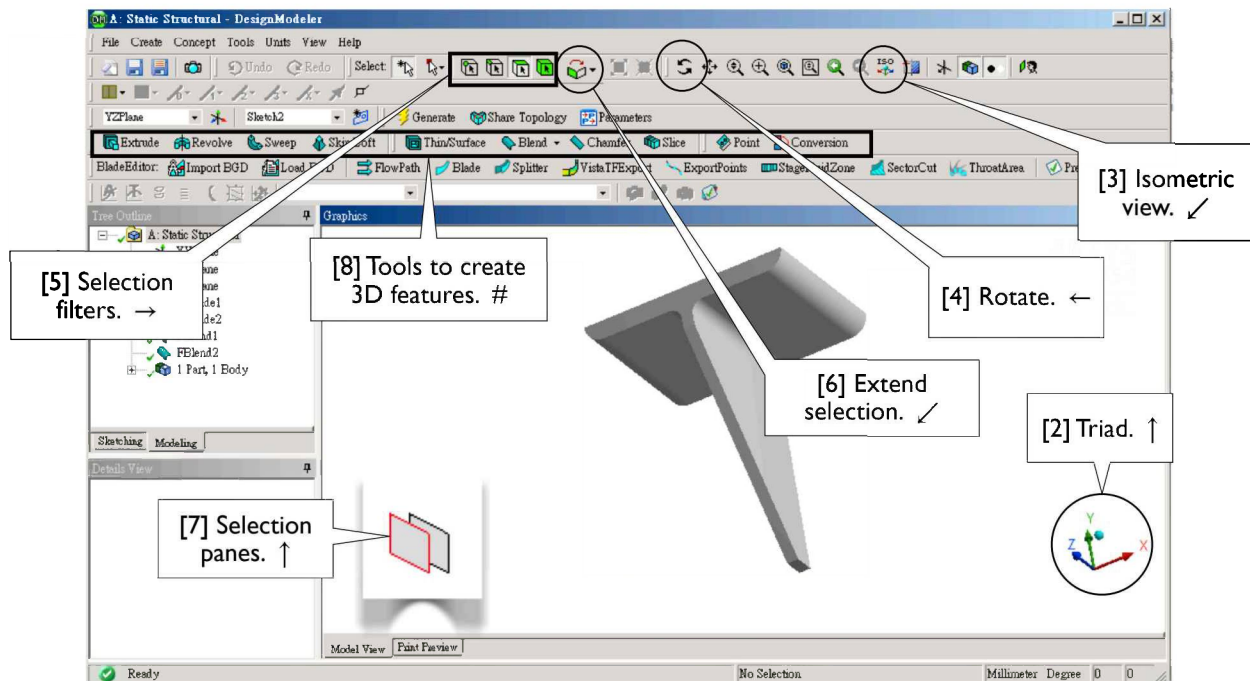
Section 4.4

More Details



4.4.1 DesignModeler GUI Revisit

[1] In Section 2.3, we've overviewed **DesignModeler GUI**, yet skipped some 3D tools, such as view orientations [2-4], and selection tools [5-7]. Also on the toolbar are tools to create 3D features [8]. These tools will be covered in this section. ↘



4.4.2 Principal Views and Isometric Views

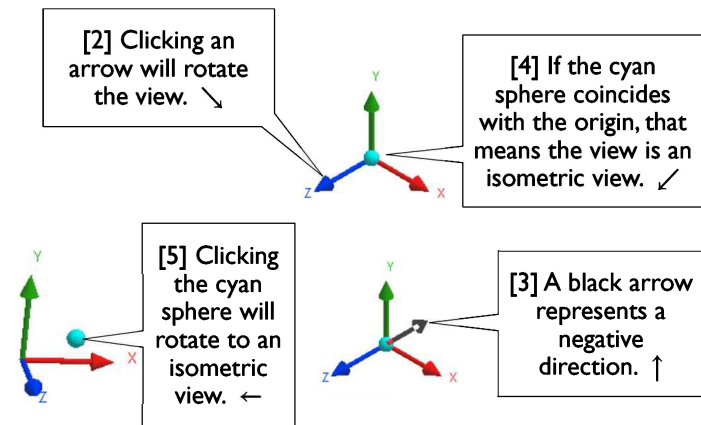
Triad^[Ref 1]

[1] On the bottom-right corner of the GUI is a triad (4.4.1 [2]); clicking an arrow will rotate the view such that the view is normal to that arrow ([2], next page). Moving the mouse over the negative side of an arrow will reveal a black arrow ([3], next page), which represents the negative direction of the arrow. You may click a black arrow to rotate the view.

Accompanying the triad is a small cyan sphere. When you rotate the view, the triad and the small cyan sphere will rotate accordingly. The small sphere represents a point located at an "isometric axis," which is a collection of points having the same magnitude of coordinates (but could be different in signs) in all three axes. Its initial position is (1, 1, 1). Thus, if the sphere coincides with the origin, that means your view is an isometric view [4]. When the sphere does not coincide with the origin, clicking the sphere will rotate to an isometric view [5]. ↙

Isometric View^[Ref 2]

[6] As mentioned, the small cyan sphere represents an isometric direction and its initial position is (1, 1, 1). In 3D space, there are a total of 8 such directions. For example: (-1, 1, 1), (1, -1, 1), etc. These are all isometric views. When you click **Isometric View** (4.4.1 [3], last page), the view will rotate to the isometric view closest to the current view. #



4.4.3 View Rotations^[Ref 3]

Rotate with Middle Mouse Button

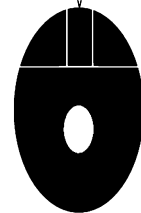
[1] Holding the middle mouse button down while moving the mouse around the graphic area, you can rotate the view [2]. It is simple and convenient. →

Rotate tool

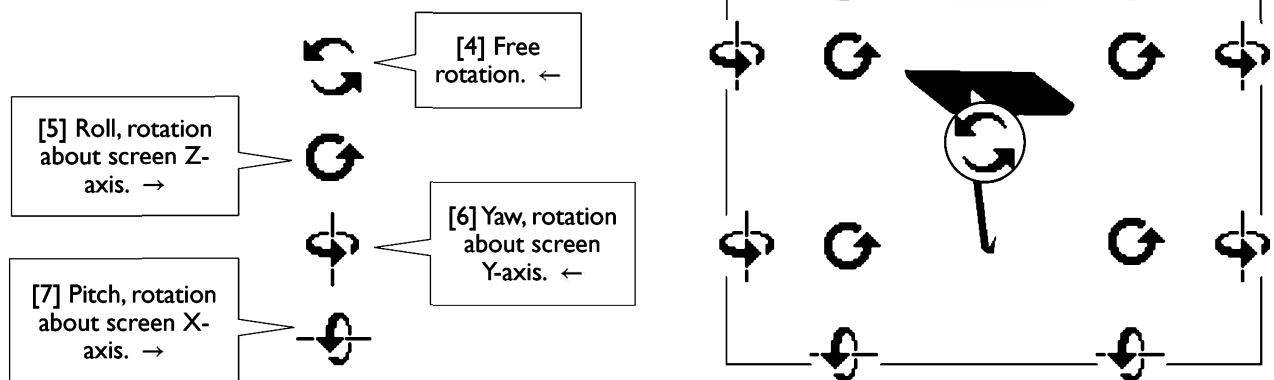
[3] The **Rotate** tool (4.4.1 [4], last page) gives you more controls than using the middle mouse button [1-2] for rotating the view. After activating the **Rotate** tool by clicking it, the mouse cursor becomes one of the four shapes [4-7], depending on the location of your mouse cursor as shown in [8]: free rotation [4] when the cursor is near the center of the graphics window; roll [5] when the cursor is away from the center; yaw [6] when the cursor is near the vertical edges; pitch [7] when the cursor is near the horizontal edges.

By default, the model center is the center of rotation. You can set the center of rotation (a red sphere) by clicking on the model. The red sphere will stay in the middle of the graphics window. To restore the center of rotation to the model center, click anywhere in the graphics window away from the model. This will re-center the model at the middle of the graphics window. ↓

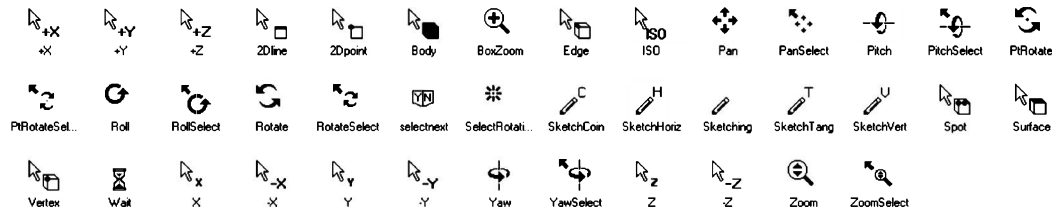
[2] Holding the middle mouse button down while moving the mouse around the graphic area, you can rotate the view. ←



[8] The type of rotation depends on the location of the cursor. #



4.4.4 Mouse Cursor



[1] The type of mouse cursor automatically changes to reflect the current operation. #

4.4.5 Selection

Selection Filters^[Ref 4]

[1] By activating a selection filter (4.4.1 [5], page 197), you can make one of four types of graphic entities (points, edges, faces, and bodies) selectable. Additional filters are available in the context menu (by right-clicking the graphic area) [2]. Multiple filters can be activated at the same time. →

Extend Selection^[Ref 4]

[3] Using the current selection as a "seed", **Extend Selection** (4.4.1 [6], page 197) allows you to include neighboring edges or faces into the selection set [4-7]. ↓



[2] Selection filters can be accessed from the context menu. ←

[4] Extend the current selection to include the adjacent tangent edges or faces. →



[5] This is equivalent to executing **Extend to Adjacent** infinite times. ←

[6] Extend the current selection to include the adjacent blend faces. →

[7] Extend the current selected faces up to the boundaries defined by selected edges. ↓

Selection Panes^[Ref 5]

[8] When you select an entity by clicking your mouse on the model, and if more than one entity lies under the mouse cursor, the graphics window displays a stack of rectangles in the lower-left corner (4.4.1 [7], page 197). The rectangles are stacked methodically with the topmost rectangle representing the most visible entity and subsequent rectangles representing entities underneath the mouse cursor, front to back. These rectangles are aliases of selectable entities, that is, highlighting and picking these rectangles are identical to that for the entities. When you move the mouse over these rectangles, the mouse cursor changes to show the type of the entity. #

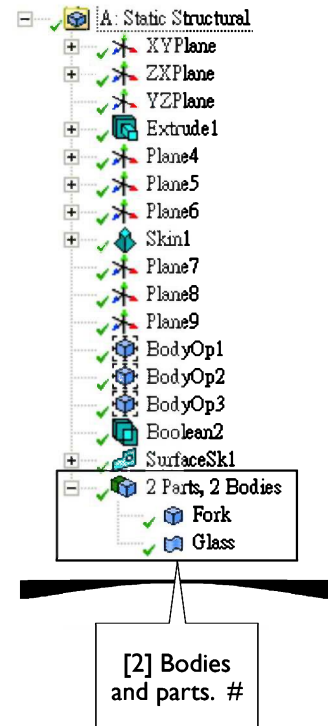
4.4.6 Bodies and Parts^[Ref 6]

[1] The last branch of the model tree contains the bodies and parts of the model [2], which are the only geometric entities that will be exported to **Mechanical** for simulations.

A body is a continuum and made of one kind of material. A 3D body is either a solid body, a surface body, or a line body.

A part consists of a single body or multiple bodies of the same type. If multiple, then all bodies are assumed to be bonded together; i.e., they form a single continuum (but multiple materials are allowed). In **Mechanical**, each body is meshed independently. If a part consists of multiple bodies, constraints are imposed on the boundaries of the contacting bodies to guarantee the compatibility between the boundaries.

A model may consist of one or more parts. Since each part is meshed independently, mesh at the boundaries between parts is not compatible. In **Mechanical**, connections^[Ref 7] (e.g., contacts, joints) among parts must be established to complete a simulation model. →



4.4.7 Feature-Based 3D Modeling

[1] A geometric model consists of features; an object in a model tree is called a feature of the model. The features include *planes*, *base features*, *placed features*, etc.

Base Features

A base feature is also called a *sketched feature* because it is created by drawing one or more sketches, and then "growing" to a 3D feature by means of **Extrude**, **Revolve**, **Sweep**, or **Skin/Loft**. A newly created base feature can add material to or subtract material from the existing bodies.

Placed Features

Some features have predefined shapes and behaviors. To add these features to existing bodies, all we have to do is to specify where we want to place these features, along with a few other settings. Therefore, these features are called *placed features*, e.g., **Blend**, **Chamfer**, **Thin/Surface**, **Slice**, etc. #

4.4.8 Base Features^[Ref 8]



[1] A base feature is created by drawing one or more sketches, and then "growing" to a 3D feature by means of **Extrude**, **Revolve**, **Sweep**, or **Skin/Loft**.

Extrude

The tool is used to extrude a sketch along its normal direction to create a 3D body. The extrusion may be symmetric or asymmetric to the sketching plane. The extrusion depth may be a fixed value, through all bodies (used only for cutting the material), up to a face, or up to a surface. A *face* is a bounded region and has a finite area while a *surface* is an unbounded region and has infinite area. A surface is often the extension of a face.

Revolve

The tool is used to revolve a sketch about an axis to create a 3D body. An angle of revolution can be specified.

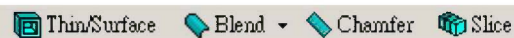
Sweep

Sweep can be thought of as a generalization of **Extrude**. The tool is used to sweep a profile along a path to create a 3D body. Both the profile and the path must be defined using sketches.

Skin/Loft

Skin/Loft can be thought of as a generalization of **Sweep**. It takes a series of profiles to create a 3D body by fitting through them. The profiles must be defined using sketches. #

4.4.9 Placed Features^[Ref 8]



[1] A placed feature has predefined shape and behavior; it can be added to an existing body.

Thin/Surface

The tool is used to convert a solid into a thin solid body or a surface body. Typically, you will select one or more faces to remove, and then specify a thickness. If the thickness is a positive value, then a thin solid body is created. If the thickness is zero, then a surface body is created.

Blend

The tool is used to create *rounds* or *fillets* on edges, or on vertices. The radius of the rounds or fillets may be fixed or variable.

Chamfer

The tool is used to create chamfer faces on edges. #

References

1. All Help>DesignModeler>ANSYS DesignModeler User's Guide>Viewing>Model Appearance Controls>Triad
2. All Help>DesignModeler>ANSYS DesignModeler User's Guide>Viewing>Rotation Modes Toolbar>Isometric View
3. All Help>Mechanical Application>Mechanical User's Guide>Application Interface>Graphic Selection>Controlling the Viewing Orientation
4. All Help>DesignModeler>ANSYS DesignModeler User's Guide>Selection>Selection Toolbar
5. All Help>DesignModeler>ANSYS DesignModeler User's Guide>Selection>Graphical Selection>Depth Picking
6. All Help>DesignModeler>ANSYS DesignModeler User's Guide>3D Modeling>Bodies and Parts
7. All Help>Mechanical Application>Mechanical User's Guide>Objects Reference>Connections
8. All Help>DesignModeler>ANSYS DesignModeler User's Guide>3D Modeling>3D Features

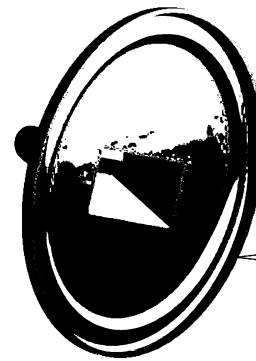
Section 4.5

LCD Display Support

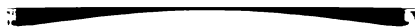
4.5.1 About the LCD Display Support

[1] The LCD Display support is made of an ABS (acrylonitrile-butadiene-styrene) plastic. The thickness is 3 mm [2]. Details of the hinge [3] are not shown on this page but will be illustrated in 4.5.4 (pages 205-206).

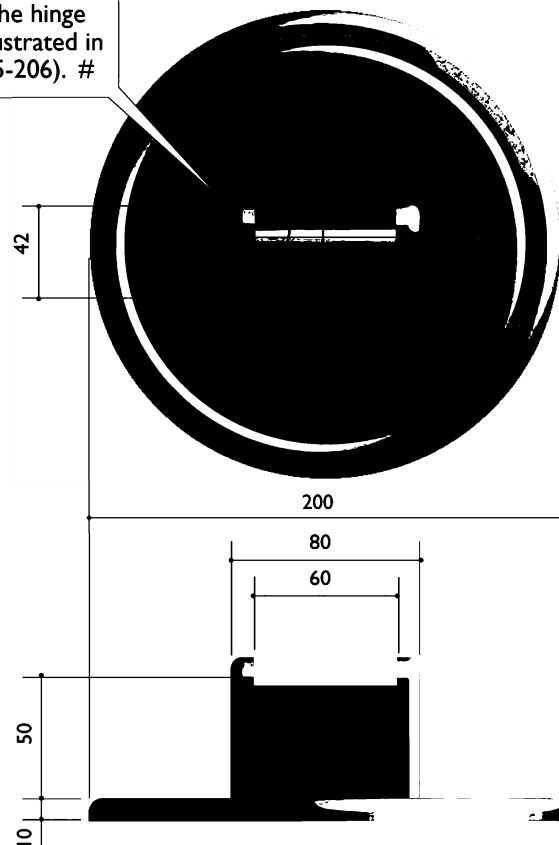
This model will be used in Section 5.4 for a static structural simulation to assess the deformation and stress under a design load. →



[2] The thickness of the plastic is 3 mm. ←

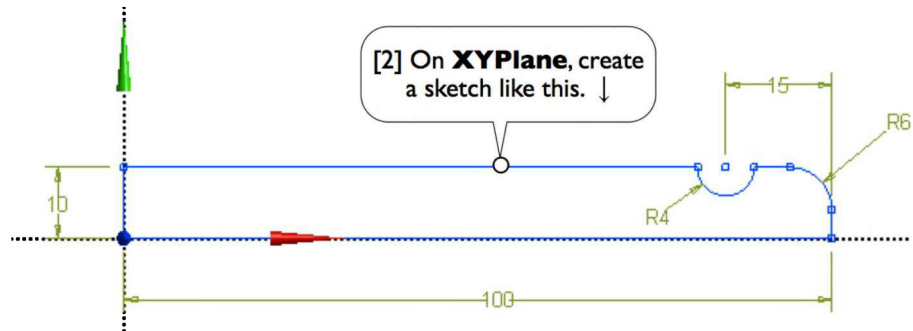


[3] Details of the hinge design will be illustrated in 4.5.4 (pages 205-206). #

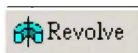


4.5.2 Create the Base

[1] Launch Workbench, create a **Static Structural** system, and save the project as **Support**. Start up DesignModeler. Select **Millimeter** as the length unit. →

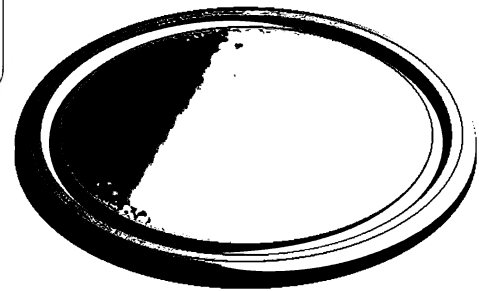


[2] On **XYPlane**, create a sketch like this. ↓

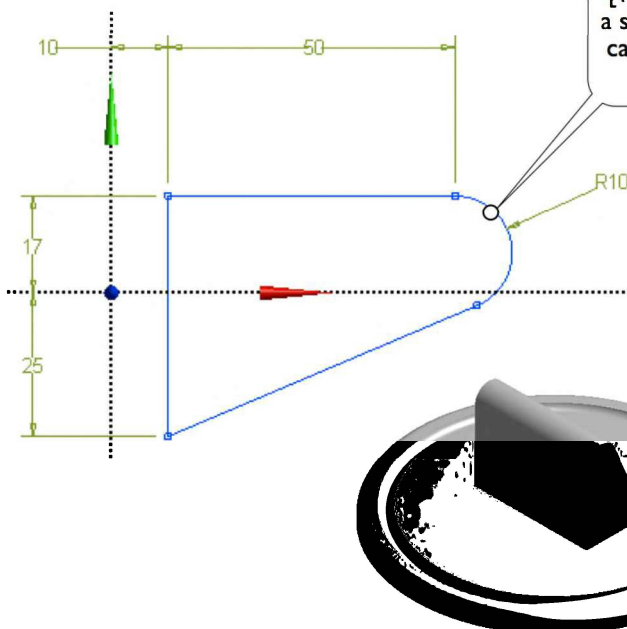


Details View	
Details of Revolve1	
Revolve	Revolve1
Geometry	Sketch1
Axis	2D Edge
Operation	Add Material
Direction	Normal
FD1, Angle (>0)	360°
As Thin/Surface?	No
Merge Topology?	Yes
Geometry Selection: 1	
Sketch	Sketch1

[3] **Revolve** the sketch to create the base. Select the vertical axis as **Axis**. #



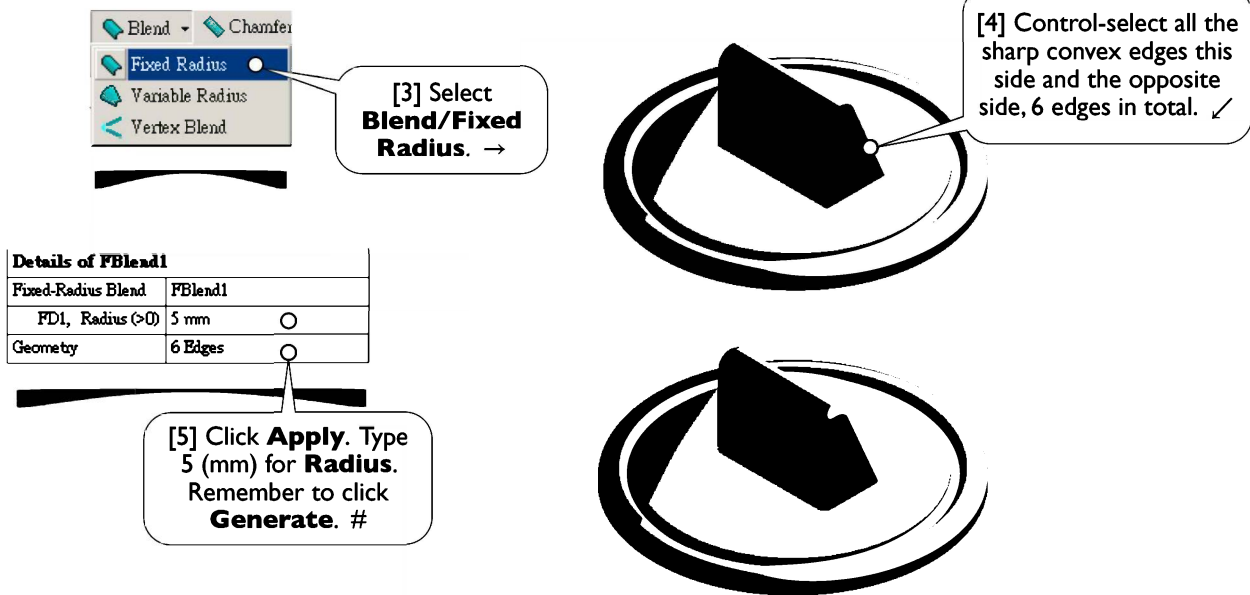
4.5.3 Create the Upholder



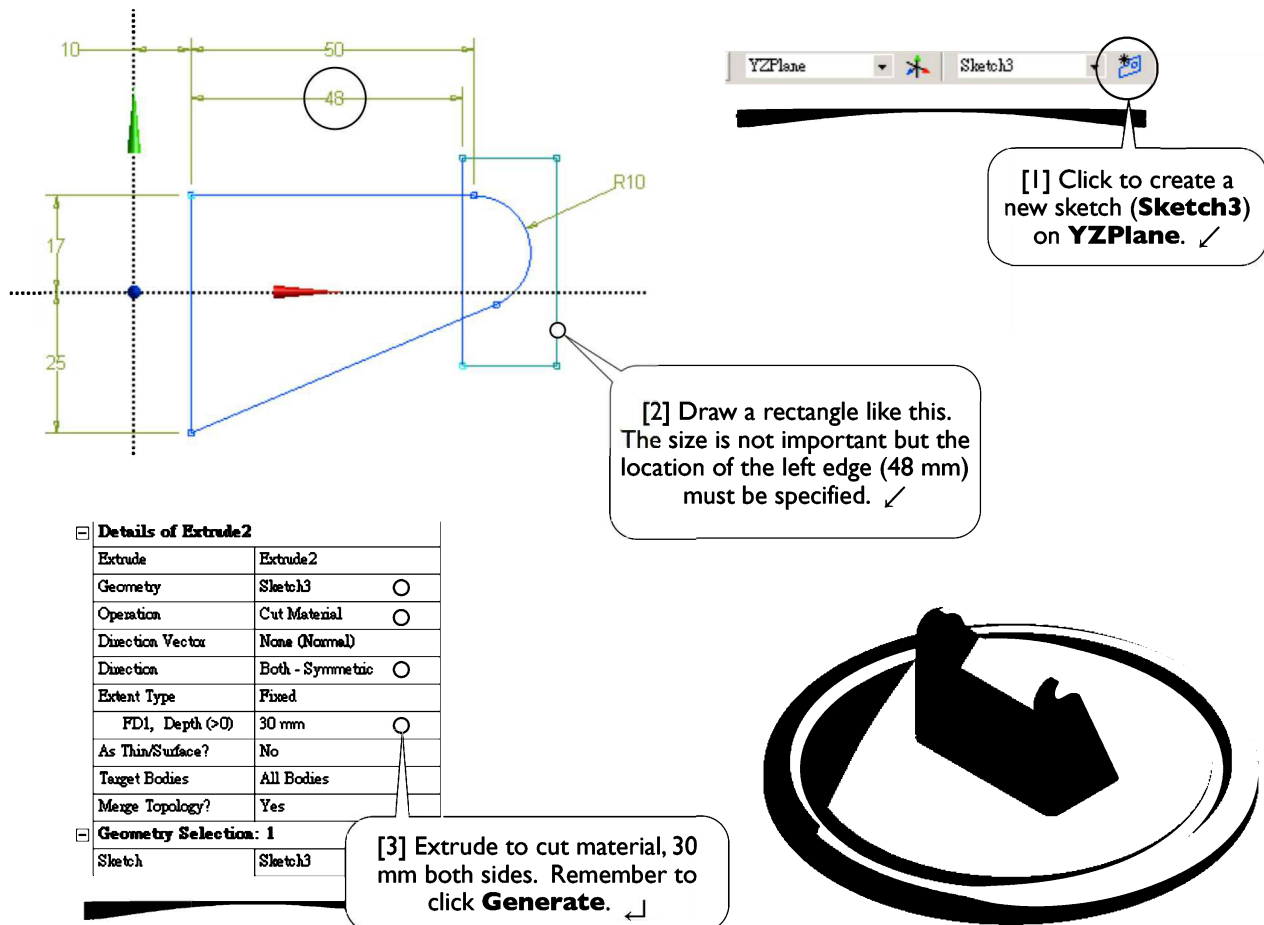
[1] On **YZPlane**, draw a sketch like this. The arc can be created using the **Fillet** tool. →

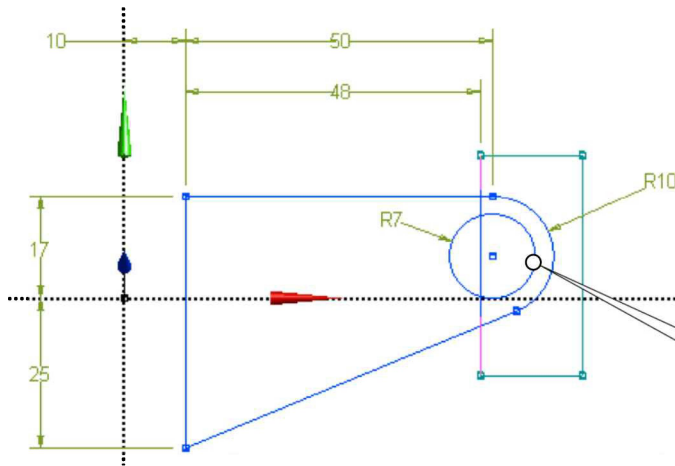
[2] Extrude 40 mm both sides. ↵

Details of Extrude1	
Extrude	Extrude1
Geometry	Sketch2
Operation	Add Material
Direction Vector	None (Normal)
Direction	Both - Symmetric
Extent Type	Fixed
FD1, Depth (>0)	40 mm
As Thin/Surface?	No
Merge Topology?	Yes
Geometry Selection: 1	
Sketch	Sketch2



4.5.4 Create Hinge



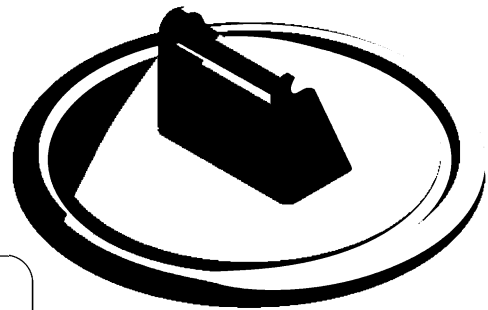


[4] Click to create a new sketch (**Sketch4**) on **YZPlane**. ✓

[5] Draw a circle and specify the radius (7 mm) like this. Impose a **Concentric** constraint for the circle and the arc. ✓

Details of Extrude3		
Extrude	Extrude3	
Geometry	Sketch4	<input type="radio"/>
Operation	Cut Material	<input type="radio"/>
Direction Vector	None (Normal)	
Direction	Both - Symmetric	<input type="radio"/>
Extent Type	Fixed	
FD1, Depth (>0)	35 mm	<input type="radio"/>
As Thin/Surface?	No	
Target Bodies	All Bodies	
Merge Topology?	Yes	
Geometry Selection: 1		
Sketch	Sketch4	

[6] Extrude to cut material, 35 mm both sides. Remember to click **Generate**. #



4.5.5 Create Thin Body

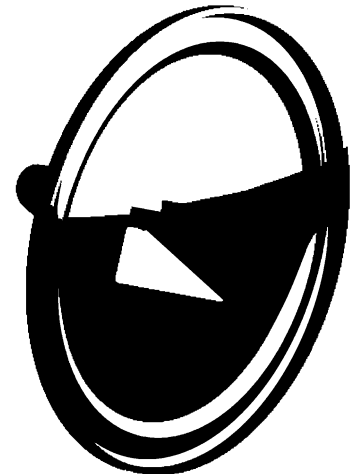
Thin/Surface



[1] Click **Thin/Surface** on the toolbar. →

Details of Thin1		
Thin/Surface	Thin1	
Selection Type	Faces to Remove	<input type="radio"/>
Geometry	1 Face	<input type="radio"/>
Direction	Inward	
FD1, Thickness (>=0)	3 mm	<input type="radio"/>
Preserve Bodies?	No	

[2] Select the bottom face to remove. Remember to click **Generate**. ↓



Wrap Up

[3] Close DesignModeler, save the project, and exit Workbench. #

Section 4.6

Review

4.6.1 Keywords

Choose a letter for each keyword, from the list of descriptions

- | | |
|---------------------------|-----------------------------------|
| 1. () Base Features | 9. () Placed Features |
| 2. () Body | 10. () Plane Outline |
| 3. () Connections | 11. () Plane Coordinate System |
| 4. () Extend Selection | 12. () Roll |
| 5. () Features | 13. () Selection Filters |
| 6. () Isotropic View | 14. () Selection Panes |
| 7. () Part | 15. () Yaw |
| 8. () Pitch | |

Answers:

1. (N) 2. (J) 3. (L) 4. (H) 5. (M) 6. (C) 7. (K) 8. (F)
 9. (O) 10. (B) 11. (A) 12. (D) 13. (G) 14. (I) 15. (E)

List of Descriptions

- (A) The coordinate system attached to a plane. Its Z-axis always points out of the plane.
- (B) A plane may be created by deriving from a face. In such cases, the plane may be designated as **Outline Plane** (i.e., a plane with boundary). You can draw entities beyond the boundary. The boundary itself is not a geometric entity. The plane boundary is used as datum edges, or as a source of replication/duplication.
- (C) A model view in which the view direction follows an isometric axis, which forms the same angles with the three principal axes.
- (D) Rotate the model about the screen Z-axis.

- (E) Rotate the model about the screen Y-axis.
- (F) Rotate the model about the screen X-axis.
- (G) A tool used to make a specific type of entity selectable. Multiple filters can be activated at the same time.
- (H) Using the current selection as seed, these tools allow you to extend the seed to include additional edges or faces into the selection set.
- (I) Rectangles appear in the lower-left corner of the graphics window. These rectangles are aliases of selectable entities, that is, highlighting and picking these rectangles are identical and synchronized for the selectable entities.
- (J) It is a continuum and made of one kind of material. A 3D body is either a solid body, a surface body, or a line body.
- (K) A collection of same type of bodies. If multiple, then all bodies are assumed to be bonded together i.e., they form a single continuum (but multiple materials are allowed). In **Mechanical**, it is meshed independently.
- (L) In **Mechanical**, they are kinematic relations between parts. Examples are contacts, joints, etc.
- (M) They are the building block of a 3D body. Examples include planes, base features, and placed features.
- (N) They are also called sketched features; they are created by first drawing one or more sketches, and then "growing" to 3D features by means of extrusion, revolution, sweeping, or lofting. A newly created one can add to or subtract material from the existing bodies.
- (O) They have predefined shapes and behaviors. To add these features to existing bodies, all we have to do is to specify where we want to place these features, along with a few other settings.

Chapter 5

3D Simulations

Many concepts and techniques in 2D simulations (Chapter 3) are applicable to 3D simulations. The problem sizes (number of nodes and elements) in 3D cases are usually larger and the geometries are usually more complicated than those in 2D cases. That implies more computing and engineering time. On the other hand, because we've been accustomed to the 3D world, the 3D simulations are more intuitive to us. As a result, newcomers often stick to 3D ways of thinking and forget that a problem often can be modeled as 2D. Remember that, if a problem can be reduced to a 2D, you have no reason to go for a 3D simulation.

Purpose of This Chapter

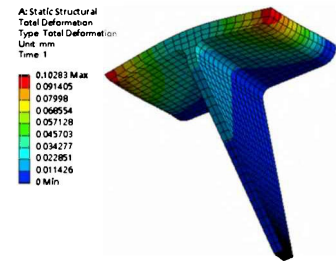
This chapter focuses on 3D simulations with solid models. Problems involving surface models and line models are discussed in Chapters 6 and 7, respectively. Like the 2D simulations in Chapter 3, this chapter focuses on linear static structural simulations. Dynamic and nonlinear simulations will be discussed in later chapters.

About Each Section

We will conduct two step-by-step examples in the first two sections. Section 5.1 is a simple example, serving as an introductory tutorial. Section 5.2 is a more involved example. Section 5.3 provides a systematic discussion trying to complement what was missed in the first two sections. Section 5.4 is an additional exercise. All exercises in this chapter use models created in Chapter 4.

Section 5.1

Beam Bracket



5.1.1 About the Beam Bracket

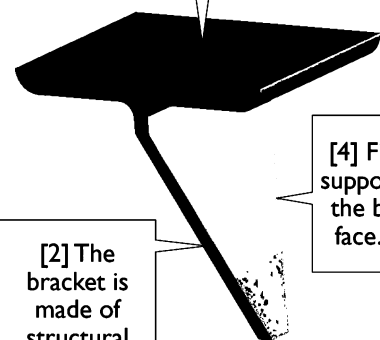
[1] In Section 4.1, we created a 3D solid model for a beam bracket. In this section, we will use the model for a static structural simulation. Besides the geometry, we need extra information for the simulation, which can be summarized into two categories: material properties and the environment conditions (i.e., loads and supports).

The beam bracket is made of structural steel [2] with a Young's modulus of 200 GPa, a Poisson's ratio of 0.3, and a yield strength of 250 MPa. The yield strength is used to assess safety factors.

The load is 27 kN uniformly distributed over the seat plate [3]. It was determined by an analysis of the entire structure.

As to the support conditions, we assume the beam bracket's back face is rigidly welded on a steel column; i.e., a fixed support [4]. →

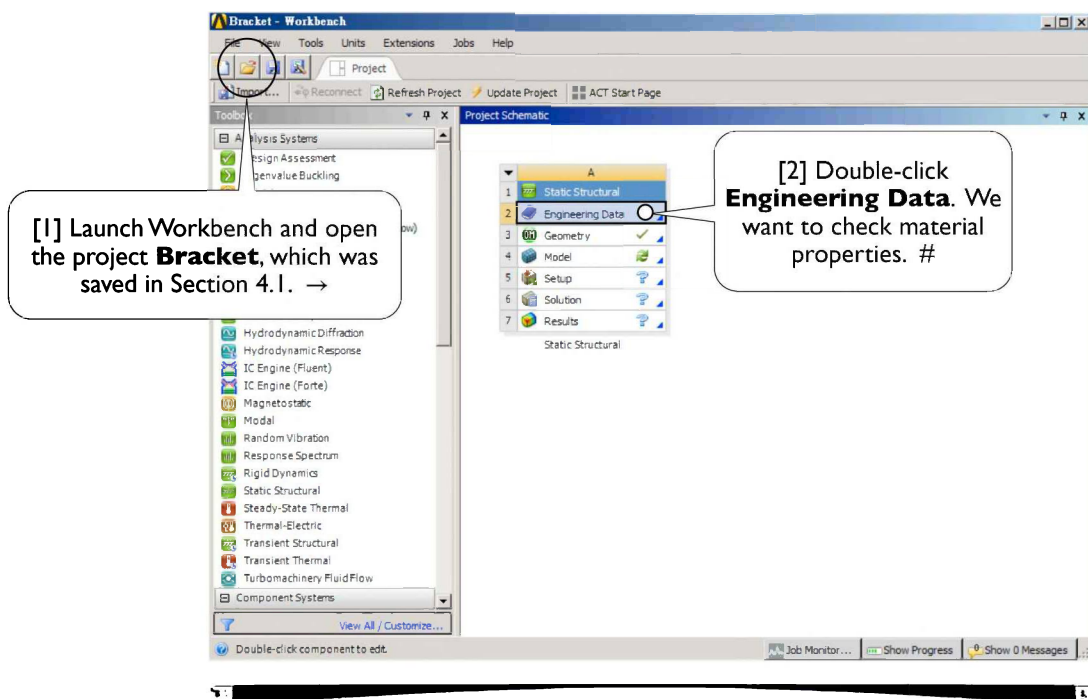
[3] The bracket is designed to withstand a load of 27 kN uniformly distributed over the seat plate. ↓



[2] The bracket is made of structural steel. ↑

[4] Fixed support at the back face. #

5.1.2 Open the Project **Bracket**



5.1.3 Check Material in **Engineering Data**

[1] Pull down **View**. Make sure **Outline** and **Properties** window panes are turned on. If they are not, select **View/Reset Workspace** to reset to a "standard" workspace. ↗

[2] Make sure **Structural Steel** is highlighted. ✓

[3] If necessary, click this plus sign to expand **Isotropic Elasticity**. ↗

[4] You may resize window panes by dragging a separator. ↓

[5] Make sure the Young's modulus is 200 GPa ($2E+11$ Pa) and the Poisson's ratio is 0.3. ↓

[6] Make sure the yield strengths are 250 MPa ($2.5E+08$ Pa). ↑

[7] Click **Project** to return to **Project Schematic**. #

Property	Value	Unit
Material Field Variables	Table	
Density	7850	kg m ⁻³
Isotropic Secant Coefficient of Thermal Expansion		
Isotropic Elasticity		
Derive from	Young's Modulus and Poisson's Ratio	
Young's Modulus	$2E+11$	Pa
Poisson's Ratio	0.3	
Bulk Modulus	$1.6667E+11$	Pa
Shear Modulus	$7.6923E+10$	Pa
Strain-Life Parameters		
S-N Curve	Tabular	
Tensile Yield Strength	$2.5E+08$	Pa
Compressive Yield Strength	$2.5E+08$	Pa
Tensile Ultimate Strength	$4.6E+08$	Pa
Compressive Ultimate Strength	0	Pa

Chart of Properties f: Young's Modulus (Pa) vs Temperature (C)

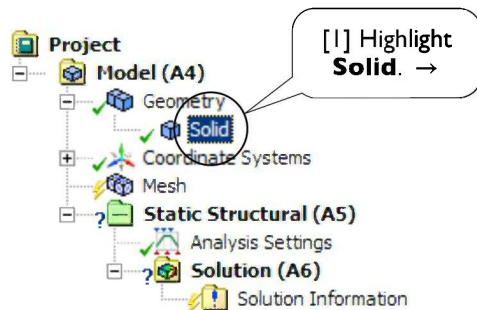
5.1.4 Start Up **Mechanical**

[1] Double-click to start up **Mechanical**. →

[2] In **Mechanical**, make sure the unit system is **mm-kg-N-s**. #

Metric (mm, kg, N, s, mV, mA)

5.1.5 Check Material Assignment



Material Assignment

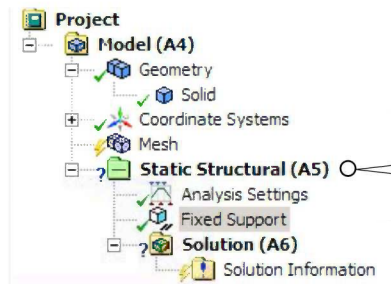
[4] By default, **Mechanical** assigns **Structural Steel** for each body; therefore, we don't need to do anything about the material assignment in this case. #

+ Graphics Properties	
- Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
- Material	
Assignment	Structural Steel
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
+ Bounding Box	
+ Properties	
+ Statistics	

[2] In the **Details**, make sure the material is **Structural Steel**, the default material. ↓

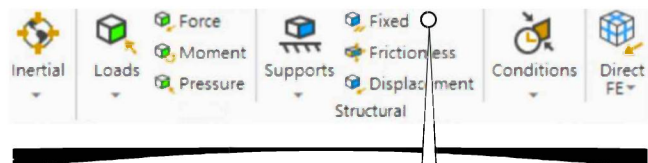
[3] Although **Nonlinear Effects** and **Thermal Strain Effects** are turned on by default, they have no effects in this case, since there are no nonlinearities or thermal loads in this case. ←

5.1.6 Specify Support

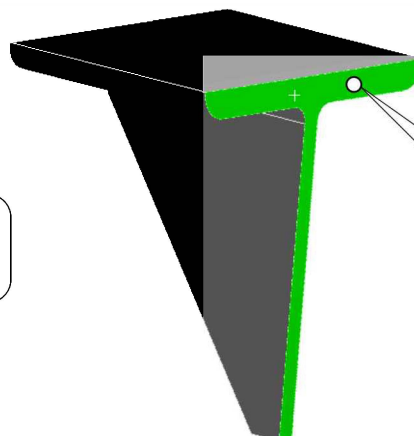


Details of "Fixed Support"	
- Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
- Definition	
Type	Fixed Support
Suppressed	No

[4] And click **Apply**. #

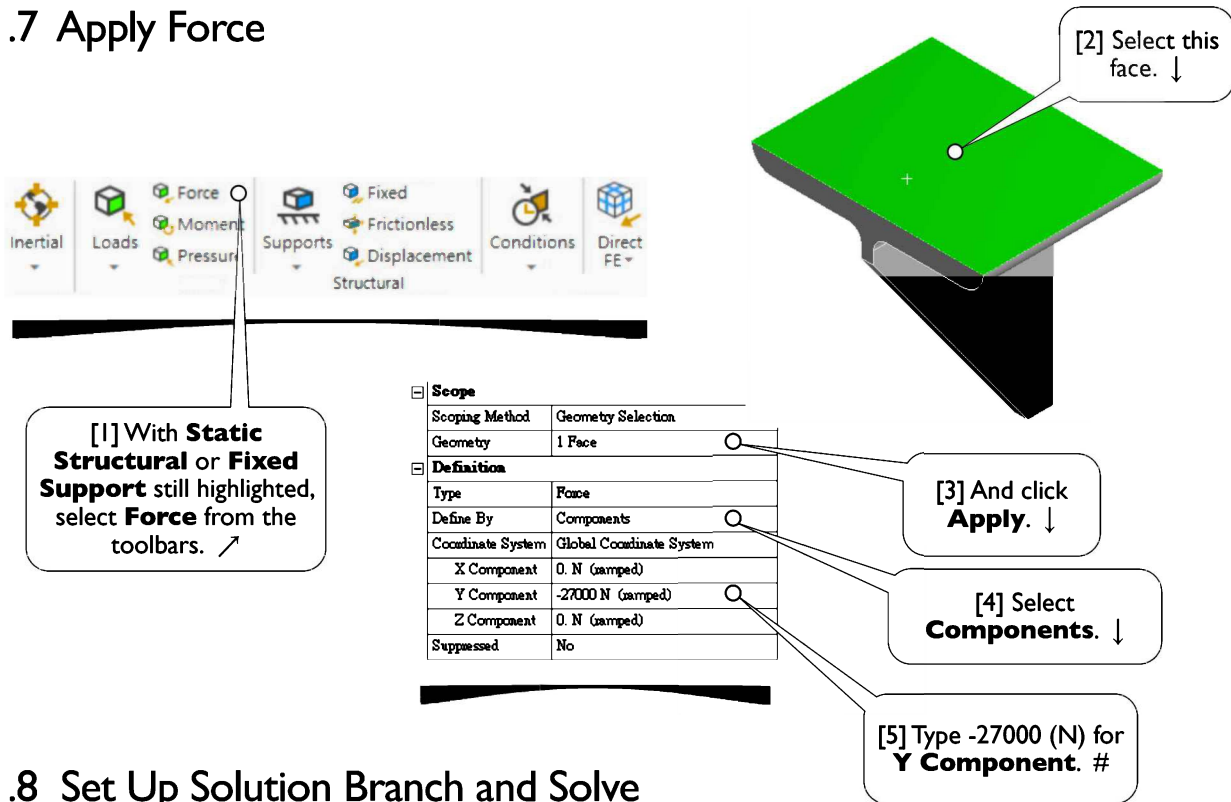


[2] From the toolbars, click **Fixed** (or right-click **Static Structural** [1] and select **Insert/Fixed Support**). ↓



[3] Rotate the model and select this face. ←

5.1.7 Apply Force



[1] With **Static Structural** or **Fixed Support** still highlighted, select **Force** from the toolbars. ↗

[2] Select this face. ↓

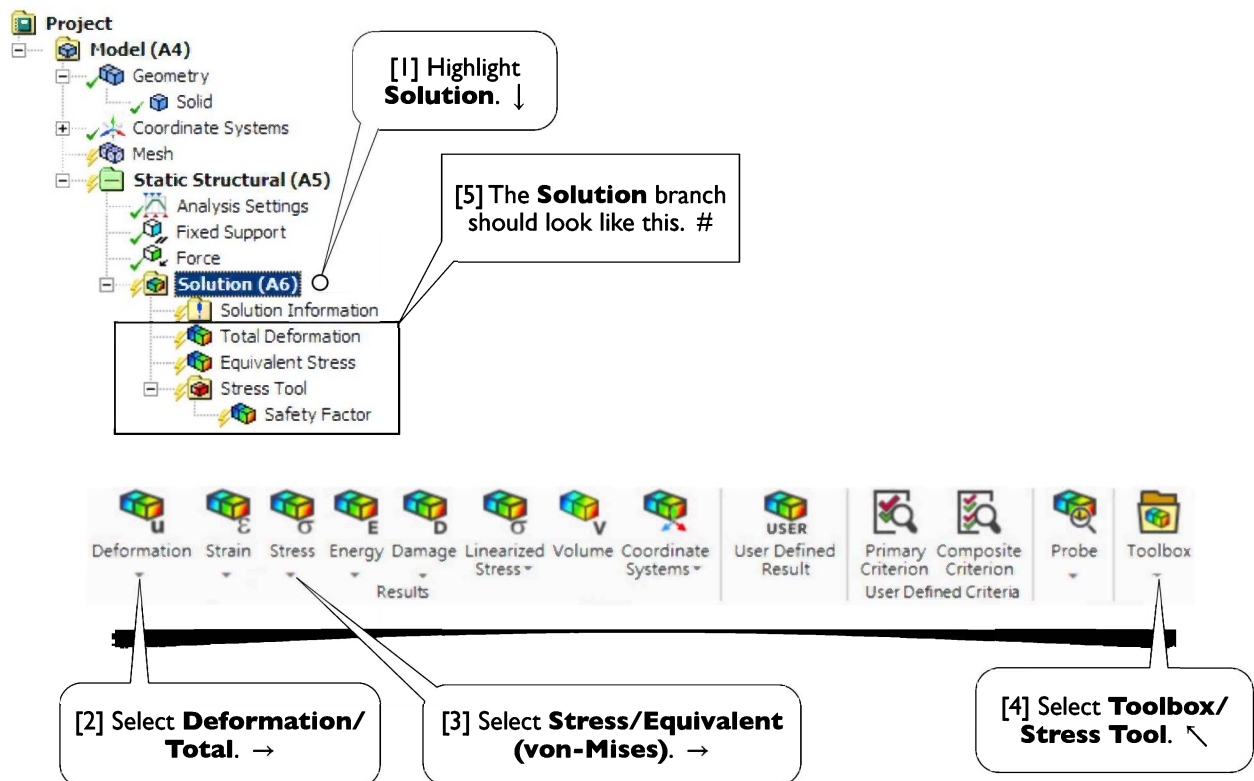
[3] And click **Apply**. ↓

[4] Select **Components**. ↓

[5] Type -27000 (N) for **Y Component**. #

Definition	
Scoping Method	Geometry Selection
Geometry	1 Face
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	0. N (samped)
Y Component	-27000 N (samped)
Z Component	0. N (samped)
Suppressed	No

5.1.8 Set Up Solution Branch and Solve



[1] Highlight **Solution**. ↓

[5] The **Solution** branch should look like this. #

[2] Select **Deformation/Total**. →

[3] Select **Stress/Equivalent (von-Mises)**. →

[4] Select **Toolbox/Stress Tool**. ↖

5.1.9 Generate Mesh and Solve the Model

Project Tree:

- Project
 - Model (A4)
 - Geometry
 - Solid
 - Coordinate Systems
 - Mesh** (highlighted)
 - Static Structural (A5)
 - Analysis Settings
 - Fixed Support
 - Force
 - Solution (A6)
 - Solution Information
 - Total Deformation
 - Equivalent Stress
 - Stress Tool
 - Safety Factor

Details of "Mesh":

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
<input type="checkbox"/> Element Size	Default
Sizing	
Use Adaptive Sizing	Yes
Resolution	4 (highlighted)
Mesh Deforming	Yes
<input type="checkbox"/> Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Diagonal	303.32 mm
Average Surface Area	7084.1 mm ²
Minimum Edge Length	10.0 mm
Quality	
Inflation	
Advanced	
Statistics	
<input type="checkbox"/> Nodes	4649
<input type="checkbox"/> Elements	2389

3D Model: A 3D model of a bracket with a mesh. The mesh is highlighted in the image.

Annotations:

- [1] Highlight **Mesh**. ↓
- [2] Expand **Sizing** and type (or select) **4** for **Resolution** (see 5.3.1, page 227) ↗
- [3] Click **Generate** from the toolbars. ✓
- [4] Expand **Statistics** to examine the mesh count. →
- [5] Click **Solve**. #

5.1.10 View the Results

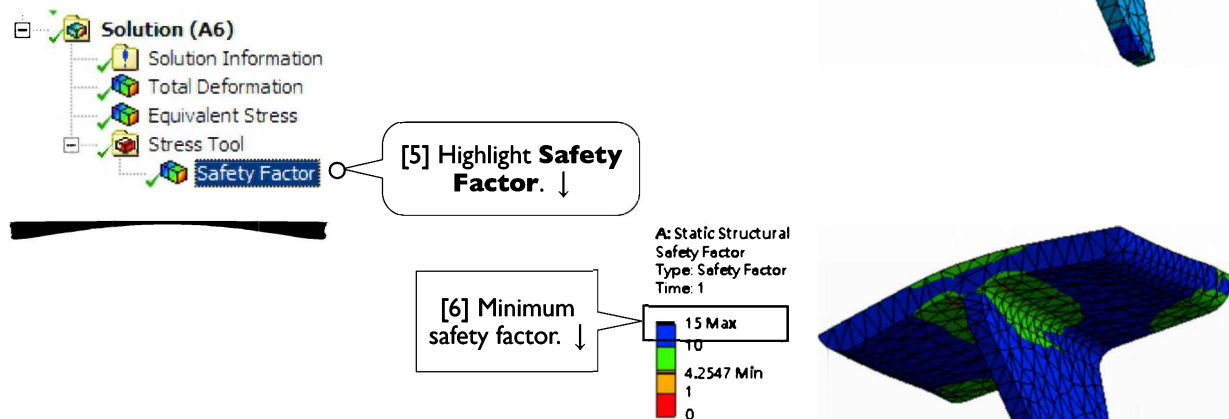
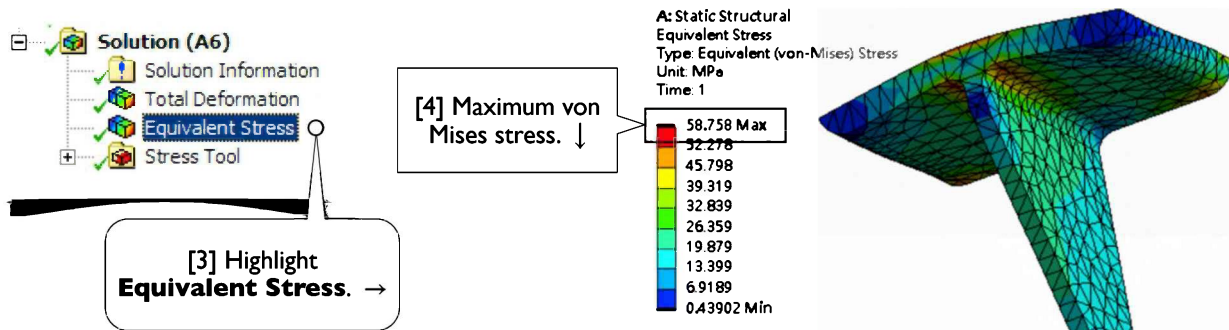
Project Tree:

- Project
 - Model (A4)
 - Geometry
 - Solid
 - Coordinate Systems
 - Mesh
 - Static Structural (A5)
 - Analysis Settings
 - Fixed Support
 - Force
 - Solution (A6)
 - Solution Information
 - Total Deformation** (highlighted)
 - Equivalent Stress
 - Stress Tool

Results: A 3D model of the bracket showing the total deformation. The color scale ranges from 0 Min (blue) to 0.10201 Max (red).

Annotations:

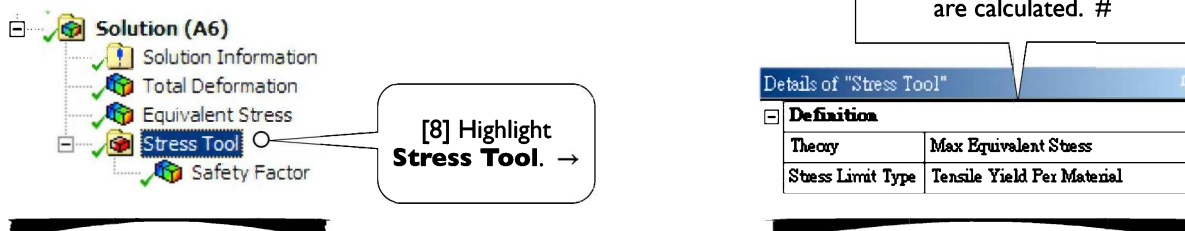
- [1] Highlight **Total Deformation**. →
- [2] Maximum deformation. ↖



Safety Factors

[7] If you highlight **Stress Tool** in the project tree [8], the details view will show you how the safety factors are calculated [9]. In this case, the safety factor is the ratio between the tensile yield strength (250 MPa, see 5.1.3[6], page 211) and the calculated equivalent stress [3-4]. For example, at the most critical region, the equivalent stress is 56.131 MPa [4]; therefore, the safety factor is 4.4539 ($= 250/56.131$) [6].

You can change the way safety factors are calculated by changing the settings in the details view [9]. ↓



5.1.11 Animate the Results

[1] Highlight **Total Deformation**. →

[2] If necessary, drag this separator (3.3.4[9], page 143) rightward to reveal more animation controls. ✓

[3] Click **Play** to animate the deformation. →

[4] Click **Stop** to stop the animation. →

[5] The **Export Video File** button allows you to save the animation in **AVI**, **MP4**, **WMV**, and **GIF** formats. #

5.1.12 Evaluate Structural Error

[1] Highlight **Solution** in the project tree and select **Stress/Error**. →

[2] A **Structural Error** object is inserted with a thunderbolt, indicating that it is not evaluated yet. ✓

[3] Click **Solve** to evaluate the **Structure Error**. ↑

[4] Maximum structural error. #

5.1.13 Improve Mesh Quality

[1] Highlight **Mesh** in the project tree and click **Method** on the toolbars (or right-click the **Mesh** and select **Insert/Method**) to insert a method under the **Mesh** branch. ↓

[2] Select the body and click **Apply**. ↓

[3] Select the **MultiZone** method. ✓

[4] Click **Generate**. ↗

[5] Highlight **Mesh** to view the mesh. The model is now meshed with all hexahedra. This is accomplished by using the **MultiZone** mesh method, which we will discuss in 5.3.2 (pages 227-228) and Chapter 9. ↓

[6] Solve. ↓

[7] Now the structural error is reduced significantly (see 5.1.12[4], last page), meaning that the solution accuracy is improved significantly. ↙

Statistics

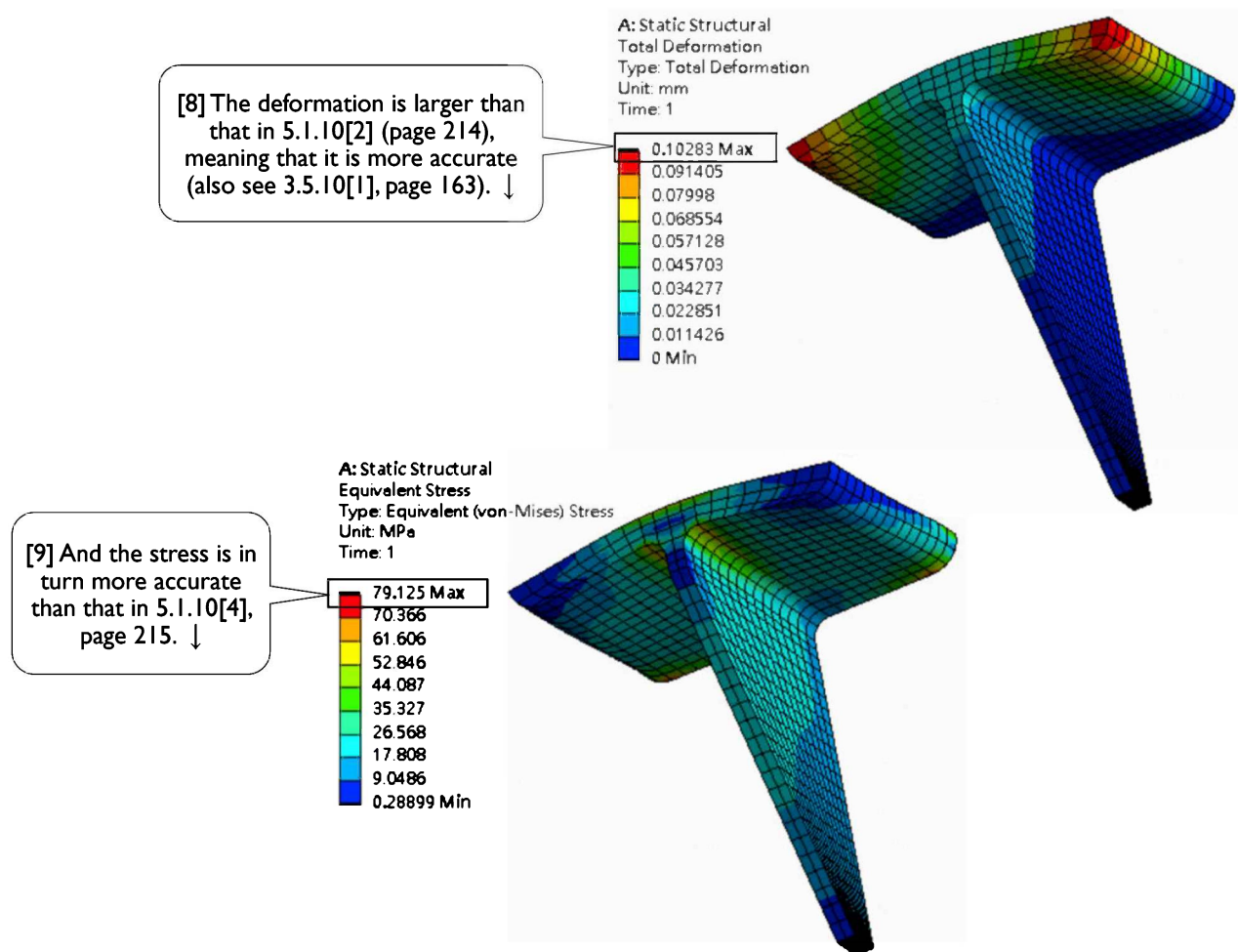
Nodes	9217
Elements	1650

Details of "MultiZone" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Not Allowed
Element Order	Use Global Setting
Size/Tig Selection	Automatic
Source Scoping Method	Program Controlled
Source	Program Controlled
Sweep Size Behavior	Sweep Element Size
Sweep Element Size	Default
Advanced	
Preserve Boundaries	Protected
Mesh Based Defeaturing	Off
Minimum Edge Length	10. mm
Write ICEM CFD Files	No

A: Static Structural
Structural Error
Type: Structural Error
Unit: mJ
Time: 1

0.080584 Max
0.07163
0.062676
0.053723
0.044769
0.035815
0.026861
0.017908
0.0089538
3.9045e-10 Min



Meshing 3D Solid Bodies

[10] Solution accuracy depends not only on mesh density but also on mesh quality. In nonlinear problems, poorer mesh quality often leads to more computing time or even failure of finding a solution.

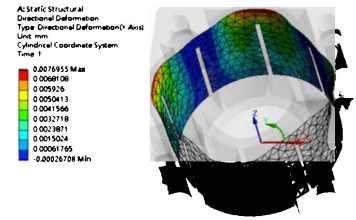
Achieving a high mesh quality is, however, not trivial. In this section, we've used a new meshing technology, namely the multi-zone method. It will be conceptually introduced in Section 5.3. Various meshing methods will be introduced in Chapter 9. ↓

Wrap Up

[11] Close **Mechanical**, save the project, and exit Workbench. #

Section 5.2

Cover of Pressure Cylinder



5.2.1 About the Cylinder Cover

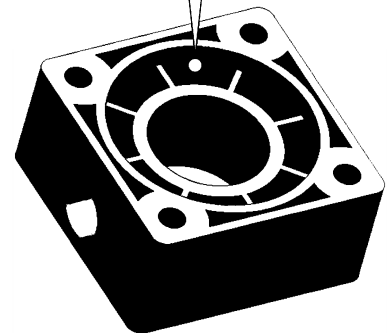
[1] In Section 4.2, we created a 3D solid model for the cover of a pressure cylinder, which is designed to hold a pressure of 0.5 MPa. In this section, we will use this model for a static structural simulation.

The cover was originally made of aluminum alloy. The purpose of the simulation is to assess the possibility of replacing the aluminum alloy with a new type of engineering plastic. The engineering plastic has a Young's modulus of 22 GPa and a Poisson's ratio of 0.3.

The engineers are concerned about the deformation, specifically the circularity of the internal surface that encompasses the pressure cylinder [2]. The circularity of a cylindrical surface can be defined as the difference between the radii of its circumscribed circle and its inscribed circle. It is required that the circularity should be less than 10 micrometers; excess of circularity beyond this value may impair the tightness and cause a leakage of gas.

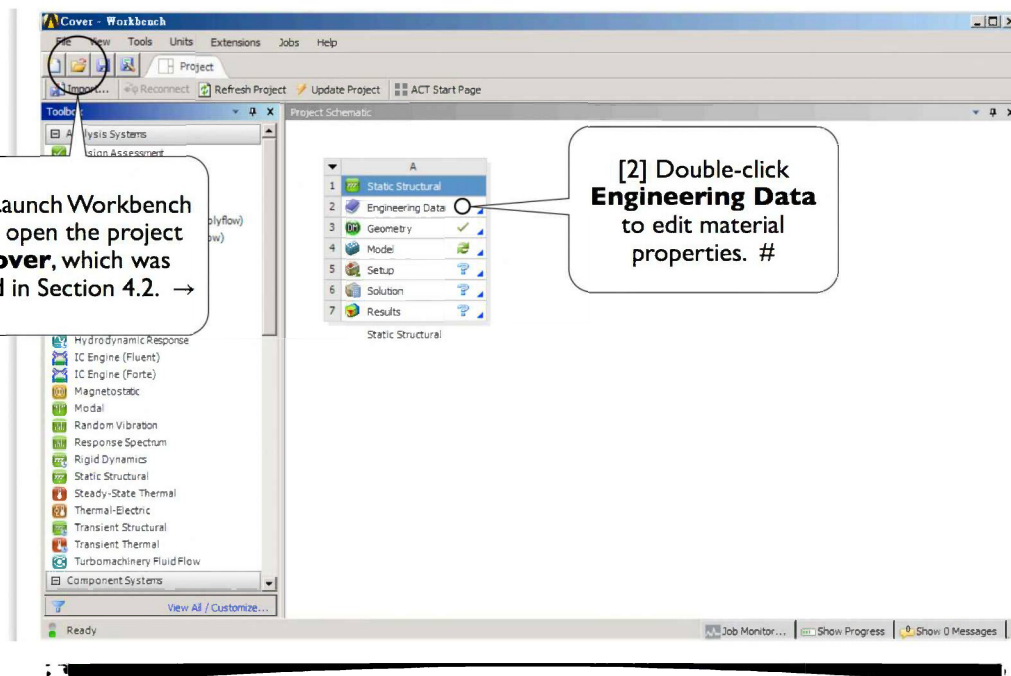
The unit system used in this exercise is **mm-kg-N-s**. →

[2] We want to investigate the circularity of this internal surface. #



5.2.2 Open the Project **Cover**

[1] Launch Workbench and open the project **Cover**, which was saved in Section 4.2. →



5.2.3 Prepare Material Properties

[1] Click here and type **Engineering Plastic** (and press **Enter**) to add a new material to **Engineering Data**. ✓

[2] In **Toolbox**, double-click **Isotropic Elasticity** to insert mechanical properties for the new material. →

[3] Type 2.2×10^{10} (Pa) for **Young's Modulus** and 0.3 for **Poisson's Ratio**. ✓

[4] **Shear Modulus** and **Bulk Modulus** are automatically calculated according to Eq. 1.2.8(2) (page 31) and Eq. 14.1.7(1) (page 532), respectively. ↓

[5] For a static structural analysis, Young's modulus and Poisson's ratio are the only two parameters to define a linearly isotropic elastic material (see Eqs. 1.2.8(1-2), page 31). Other material properties, such as shear modulus and bulk modulus, have default values that are calculated from the Young's modulus and Poisson's ratio. Besides, if thermal loads (i.e., temperature changes) are involved in the simulation, the coefficient of thermal expansion (which can be found in **Physical Properties** of **Toolbox** [6]) must also be included (see Eq. 1.2.8(3), page 32). ↑

[6] **Physical Properties**. ✓

[7] Click **Project** to return to **Project Schematic**. ↓

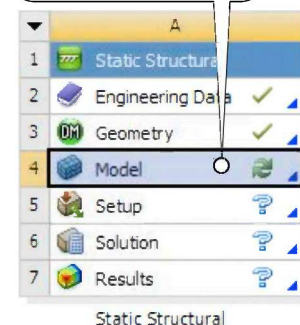
[8] Double-click to start up **Mechanical**. #

	A	B	C	D
1	Contents of Engineering Data		Source	Description
2	Material			
3	Structural Steel			Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material			

	A	B	C	D	E
1	Property	Value		Unit	
2	Isotropic Elasticity				
3	Derive from	Young's Modulus and Poisson's Ratio			
4	Young's Modulus	2.2E+10		Pa	
5	Poisson's Ratio	0.3			
6	Bulk Modulus	1.8333E+10		Pa	
7	Shear Modulus	8.4615E+09		Pa	

What material properties are needed?

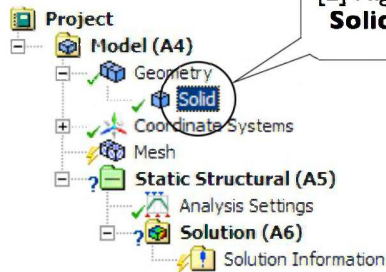
[5] For a static structural analysis, Young's modulus and Poisson's ratio are the only two parameters to define a linearly isotropic elastic material (see Eqs. 1.2.8(1-2), page 31). Other material properties, such as shear modulus and bulk modulus, have default values that are calculated from the Young's modulus and Poisson's ratio. Besides, if thermal loads (i.e., temperature changes) are involved in the simulation, the coefficient of thermal expansion (which can be found in **Physical Properties** of **Toolbox** [6]) must also be included (see Eq. 1.2.8(3), page 32). ↑



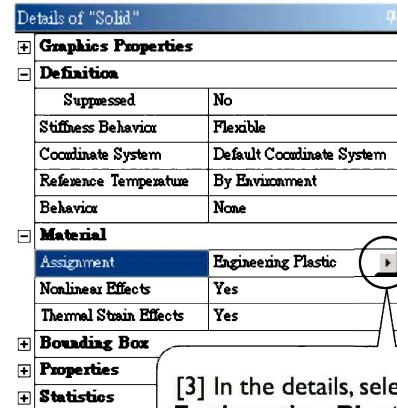
5.2.4 Assign Material to the Body

[1] Make sure the unit system is **mm-kg-N-s**. ↓

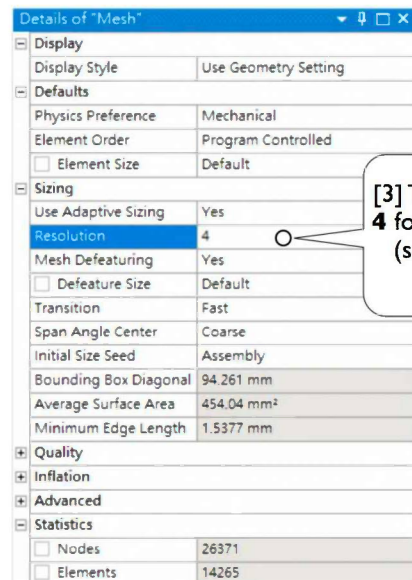
▲ Metric (mm, kg, N, s, mV, mA)



[2] Highlight **Solid**. →



[3] In the details, select **Engineering Plastic** for **Assignment**. #

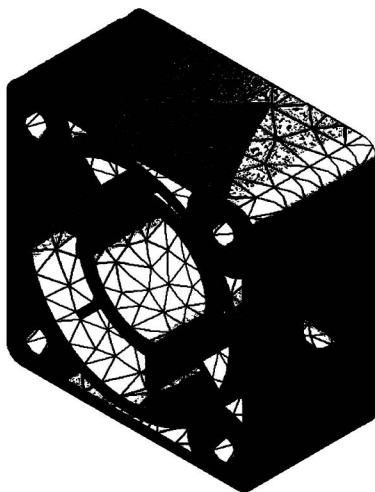


[3] Type (or select) **4** for **Resolution** (see 5.3.1, page 227). ←

5.2.5 Generate Mesh

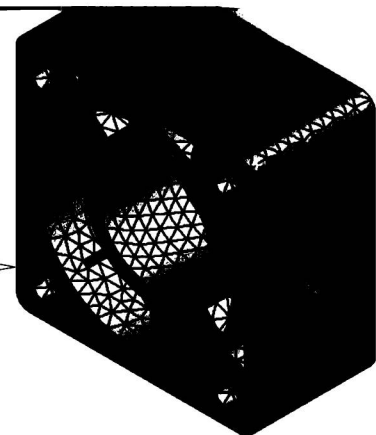


[1, 4] With **Mesh** highlighted, click **Generate**. ↓

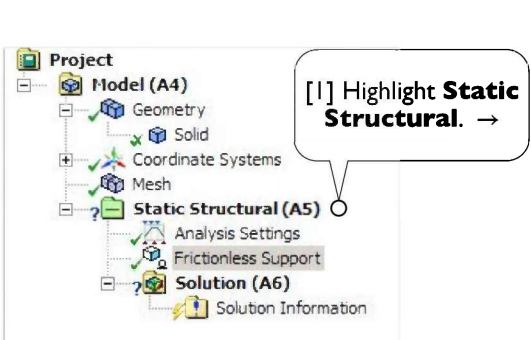
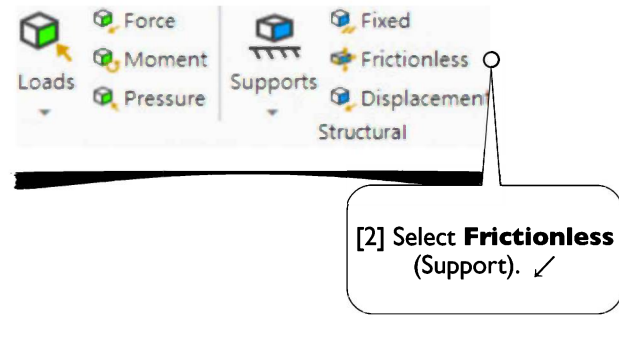
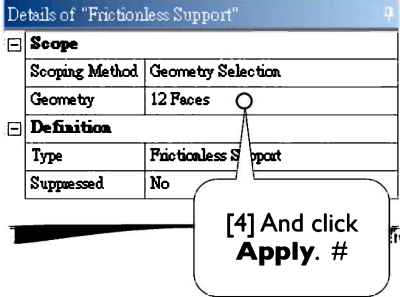
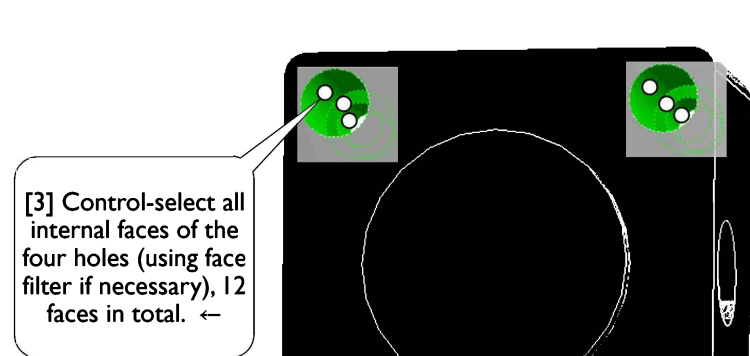


[2] The default settings generate about 5700 elements and 11,000 nodes. ↗

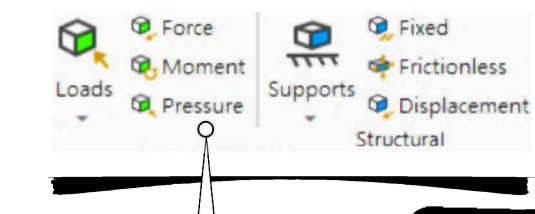
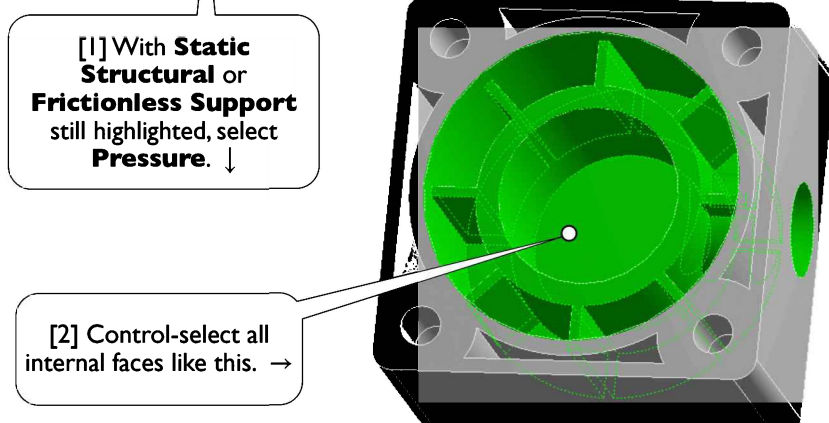
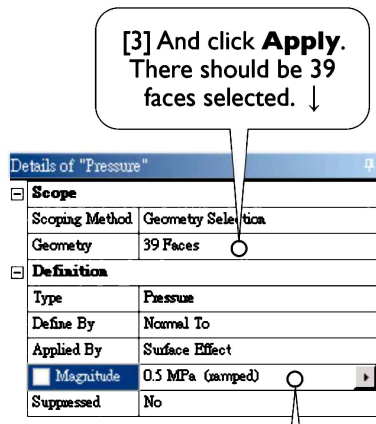
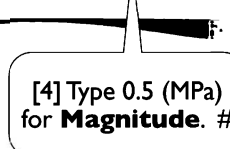
[5] The new mesh consists of about 14,000 elements and 26,000 nodes. We'll use this mesh. #



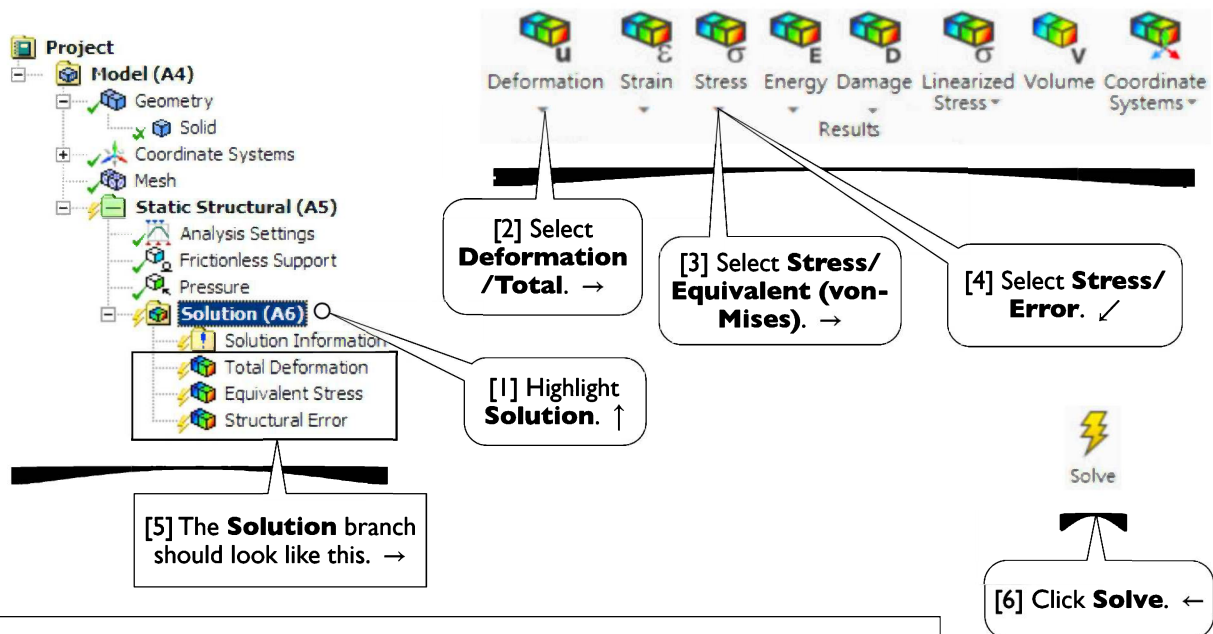
5.2.6 Specify Supports

5.2.7 Apply Pressure

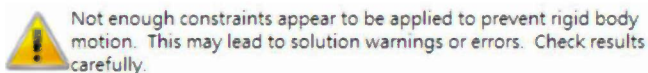





5.2.8 Set Up Solution Branch and Solve



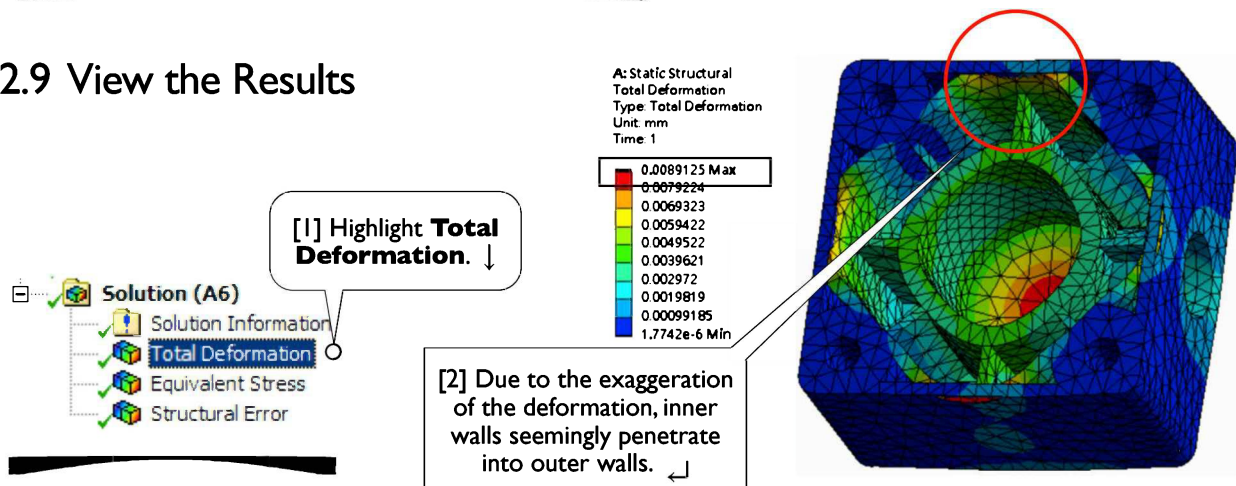
Not Enough Constraints?

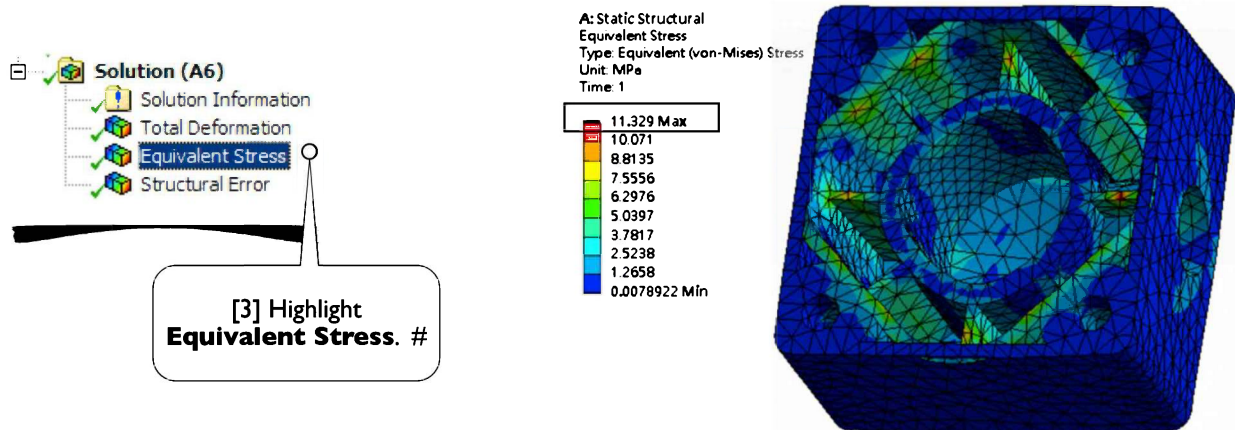
[7] After solving the model, a warning message shows up [8]. If you carefully examine the support conditions, the rigid body motions in all directions have been constrained by the frictionless supports (5.2.6, last page). Ignore the warning message; it might be a bug. ↓



[8] A warning message after solving the model. #

5.2.9 View the Results





5.2.10 Create a Cylindrical Coordinate System

The figure illustrates the steps to create a cylindrical coordinate system:

- [1] Highlight Coordinate Systems.** ↑ The 'Coordinate Systems' folder is highlighted in the Project tree.
- [2] Click **Coordinate System** to create a coordinate system.** → The 'Coordinate System' button is clicked in the 'Tools' palette.
- [3] Select **Cylindrical**.** ↓ The 'Cylindrical' option is selected in the 'Details of "Coordinate System"' dialog box.
- [4] Select **Global Coordinates** for defining the origin. Leave other settings their default values.** ✓ The 'Global Coordinates' option is selected under 'Define By' in the 'Details of "Coordinate System"' dialog box.
- [5] A cylindrical coordinate system is created. Workbench always uses X-Y-Z for the coordinate axes. For a cylindrical coordinate system, they should be interpreted as R-Theta-Z. →** A diagram shows a cylindrical coordinate system with axes labeled X, Y, and Z. A callout box explains: [5] A cylindrical coordinate system is created. Workbench always uses X-Y-Z for the coordinate axes. For a cylindrical coordinate system, they should be interpreted as R-Theta-Z. →
- [6] Right-click and rename the newly created coordinate system as **Cylindrical Coordinate System**.** # The 'Cylindrical Coordinate System' is renamed in the Project tree.

5.2.11 Assess Circularity

[1] Highlight **Solution.** ✓

[2] Select **Deformation/Directional.** →

[3] Click here to bring up **Apply/Cancel** buttons. ↑

[4] Select this cylindrical face. ↘

[5] And click **Apply.** ↓

[6] Select **Cylindrical Coordinate System**, created in 5.2.10, last page. ↑

[7] In a cylindrical coordinate system, the X-direction now should be interpreted as the radial-direction. ←

[8] Click **Solve** to evaluate the radial displacement. ↘

[9] Here, the radial displacement is 7.70 micrometers outward. ←

[10] Here, the radial displacement is 0.27 micrometers inward. ↙

Details of "Directional Deformation"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Directional Deformation
Orientation	X Axis
By	Time
Display Time	Last
Coordinate System	Cylindrical Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
Results	
Information	

A: Static Structural
Directional Deformation
 Type: Directional Deformation(X Axis)
 Unit: mm
 Cylindrical Coordinate System
 Time: 1

0.0076955 Max
 0.0068108
 0.005926
 0.0050413
 0.0041566
 0.0032718
 0.0023871
 0.0015024
 0.00061765
 -0.00026708 Min

Conclusions

[11] The circularity of the cover under pressure, according to its definition, is 7.97 micrometers ($7.70 + 0.27$; see 5.2.11 [9, 10], last page). It is within the spec requirement (10 micrometers).

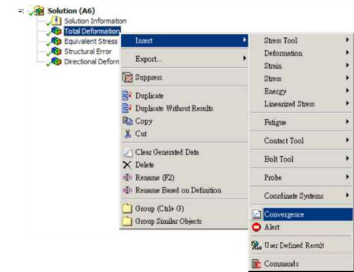
The engineers need not worry about the stress, since the stress (11.329 MPa; see 5.2.9 [3], page 224) is well below the fracture stress (which is about 50+ MPa). ↓

Wrap Up

[12] Close **Mechanical**, save the project, and exit Workbench. #

Section 5.3

More Details



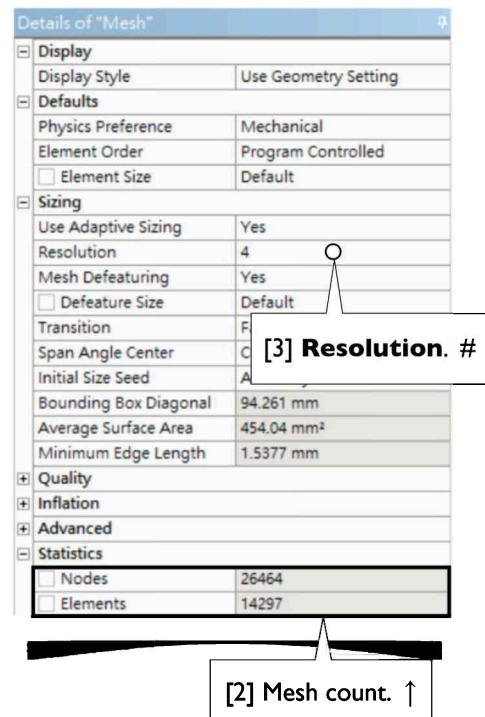
5.3.1 Global Mesh Controls^[Ref 1]

[1] By highlighting **Mesh** in a project tree, you can access the details of **Mesh**.

A statistics of the mesh count shows up at the bottom of the details [2]. The mesh count, number of nodes/elements, provides an estimation of the problem size. In 3D cases, as mentioned, the total degrees of freedom equal three times the number of nodes (1.3.1 [2], page 35). In the case of Section 5.2 (see [2]), Workbench solves a system of equations of degrees of freedom 79,392 (26,464x3, see [2]). The matrix $[K]$ has a size of 79,392x 79,392!

Resolution [3] provides a way of global mesh control. The range of values that can be set is 0 (coarse) to 7 (fine). A value of -1 will set the **Resolution** to the default value; when the **Physics Preference** is **Mechanical**, the default value is 2.

→



5.3.2 Mesh with MultiZone Method

[1] Generally, hexahedral elements are more desirable than other shapes, such as tetrahedral. The main reason is that hexahedral (or quadrilateral in 2D cases) has better convergence behavior (3.5.10, page 163; 9.3.13 and 9.3.14, page 361). That implies, with the same problem size, the hexahedral (or quadrilateral in 2D cases) gives more accurate results. Besides the shapes, mesh quality is also a key factor affecting convergence behavior. A mesh of hexahedral elements with poor mesh quality might be less desirable than tetrahedral with good mesh quality. Mesh quality will be discussed in Chapter 9.

For 2D models, Workbench usually does a good job and meshes them with all-quadrilateral elements. For example, the models in Sections 3.1 and 3.2 are meshed with quadrilateral elements without further mesh controls.

For 3D models, meshing is much more challenging. In 5.2.5 (page 221), the geometry is relatively complicated, so Workbench chooses to mesh with all-tetrahedra. ↩

[2] A simple idea to create hexahedral elements is to mesh faces of a body with quadrilaterals and then "sweep" along a depth up to other end faces of the body. The starting faces are called **source faces** and the ending faces are **target faces**. The source or target faces can be either manually or automatically selected. However, many bodies are not sweepable. The idea of the **MultiZone** method is to decompose a non-sweepable body into several sweepable bodies, and then apply **Sweep** method on each of the bodies. This is what we had done in 5.1.13[1-5] (page 217), where we inserted a **MultiZone** method and the Workbench decomposed the body into several sweepable bodies and easily "swept" each body with hexahedral elements. The result is an all-hexa mesh. #

5.3.3 Coordinate Systems^[Ref 2]

[1] When defining an environment condition or a solution object by **Components**, you need to refer to a coordinate system. By default, **Global Coordinate System** is used, which is a Cartesian coordinate system. Sometimes this coordinate system is not convenient. In such cases, we may define additional coordinate systems.

To define a coordinate system, you need to define the type of the coordinate system [2], the origin [3], and the axes [4].

Currently, workbench supports only two types of coordinate systems: **Cartesian** and **Cylindrical** [2].

Defining the origin is straightforward. You can click a location or type the coordinates [3].

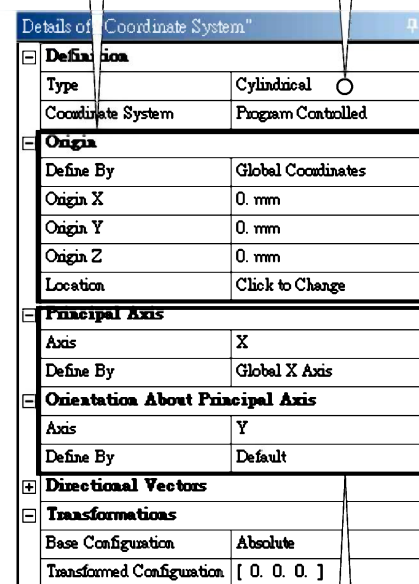
There are many ways to define the axes. Basically, you need to define two of the three axes and the third axis is automatically defined according to the right-hand rule.

The first axis you define is called the **Principal Axis** and the second axis is defined by **Orientation About Principal Axis**.

For cylindrical coordinate system, you may be confused by such terminology. Fortunately, a triad always appears on the graphics window (e.g., 5.2.10[5], page 224) and you should be visually aware if you make mistakes. ↗

[3] Definition of the origin. ↓

[2] Type of coordinate systems. ←



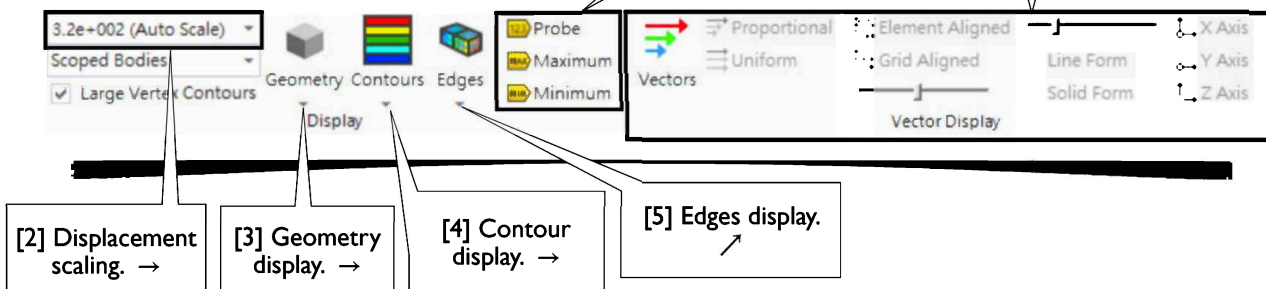
[4] Definition of the axes. #

5.3.4 Results View Controls^[Ref 3]

[1] With a solution object highlighted, the toolbar displays tools that can be used to control how the results are displayed [2-7]. ↓

[7] Click to attach a tag on the model. #

[6] Vector display. ←



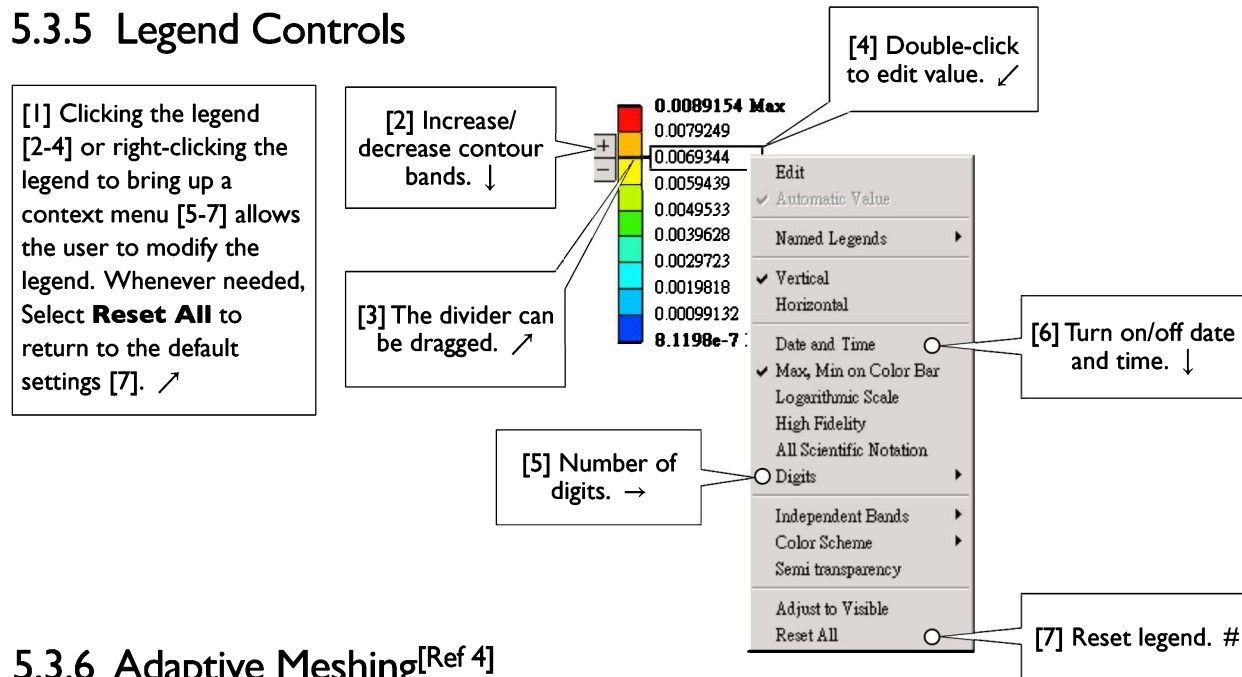
[2] Displacement scaling. →

[3] Geometry display. →

[4] Contour display. →

[5] Edges display. ↗

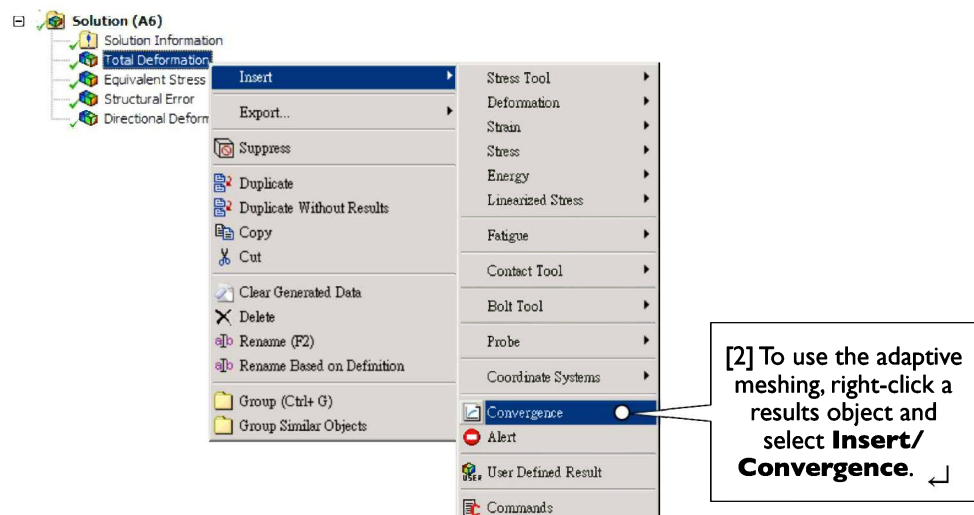
5.3.5 Legend Controls

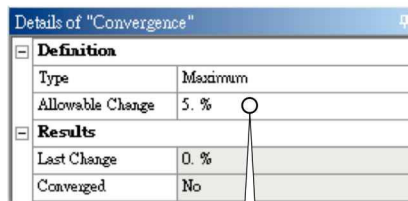


5.3.6 Adaptive Meshing^[Ref 4]

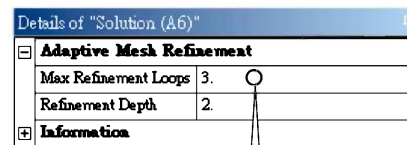
[1] We demonstrated a procedure of finite element convergence study in PART C of Section 3.5 (pages 161-163). Given an error, say 5%, we may refine mesh until the accuracy reaches this level. Performing these tasks manually is cumbersome. Workbench provides a tool to automate the mesh refinement until a user-specified level of accuracy is reached. This idea is termed *adaptive meshing*. Internally, Workbench uses structural errors (3.5.7, page 161) to help adjust the mesh, that is, it refines the mesh size in the area of large structural errors.

To use this tool, right-click a results object and select **Insert/Convergence** [2]. In the details view, specify the accuracy, or **Allowable Change** ([3], next page). Also, highlight **Solution** in the project tree and, in the details view, specify the maximum number of mesh refinement loops ([4], next page). When solving the model, Workbench will iterate to refine the mesh until the difference between two iterations is less than the **Allowable Change** or the **Max Refinement Loops** is reached. ↓





[3] In the details view, specify the accuracy (**Allowable Change**). →



[4] Highlight **Solution** and, in the details view, specify the maximum number of mesh refinement loops. #

References

1. All Help>Meshing>Meshing User's Guide>Global Mesh Controls
2. All Help>Mechanical Application>Mechanical User's Guide>Objects Reference>Coordinate System
3. All Help>Mechanical Application>Mechanical User's Guide>Steps for Using the Application>Review Results
4. All Help>Mechanical Application>Mechanical User's Guide>Objects Reference>Convergence

Section 5.4

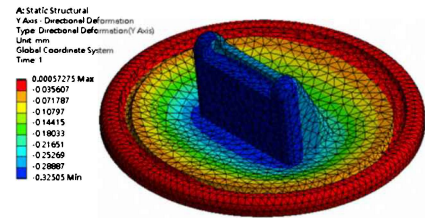
LCD Display Support

5.4.1 About the LCD Display Support

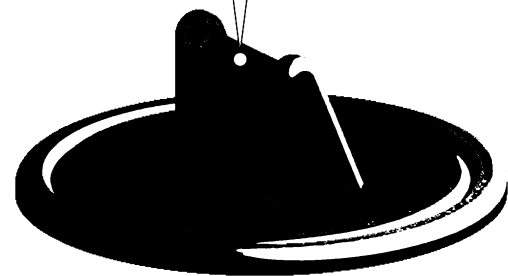
[1] The geometry created in Section 4.5 is used in this section for a static structural simulation to assess the deformation and stress under a design load [2].

The display support is made of an ABS (acrylonitrile-butadiene-styrene) plastic; its Young's modulus is 2.62 GPa and its Poisson's ratio is 0.34. In a tensile test, the material starts to develop fine cracks at 37 MPa, and fractures at 54 MPa.

The support is designed for a 17" LCD display, which weighs 40 N and is used as the design load. →



[2] The load applies on the trough. #



5.4.2 Prepare Material Properties

[1] Launch Workbench and open the project **Support**, which was saved in Section 4.5. Double-click **Engineering Data**. Add a material to **Engineering Data** [2] and specify the Young's modulus and Poisson's ratio [3-4].

Return to **Project Schematic** [5] and double-click **Model** to start up **Mechanical**. In **Mechanical**, make sure the unit system is **mm-kg-N-s**, and assign the newly created material to the solid body [6]. ✓

[5] Click to return to **Project Schematic**. ↘

[3] Double-click **Isotropic Elasticity**. →

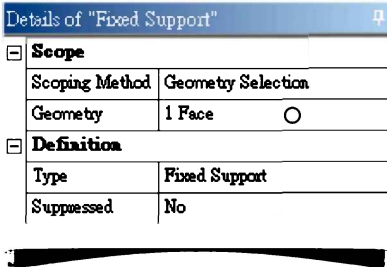
[2] Click and enter **ABS (PA-757)** to add a new material to **Engineering Data**. ↑

[4] Type values for the Young's modulus and Poisson's ratio. ↑

Property	Value
Isotropic Elasticity	
Derive from	Young's Modulus and Poisson's Ratio
Young's Modulus	2.62E+09 Pa
Poisson's Ratio	0.34
Bulk Modulus	2.7292E+09 Pa
Shear Modulus	9.7761E+08 Pa

[6] In **Mechanical**, assign the new material to the solid body. #

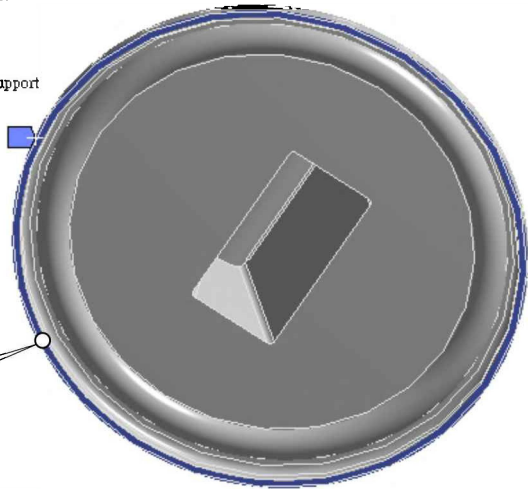
5.4.3 Specify Support



[1] Specify **Fixed Support** for the bottom rim face. Note that it would be more realistic if we specify **Compression Only Support** for the bottom rim face. For this case, however, the calculated results would be the same, since the rim face will not separate from the surface that supports the model. A **Compression Only Support** would introduce contact nonlinearity; that's why we avoid it. #

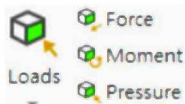
A: Static Structural
Fixed Support
Time: 1. s

Fixed Support

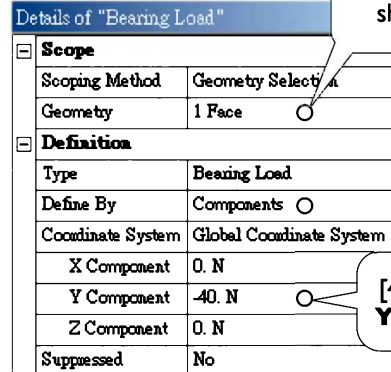


[2] Select the cylindrical surface as shown in [3]. ↓

5.4.4 Specify Load



[1] Select **Loads/ Bearing Load** from the toolbars. →

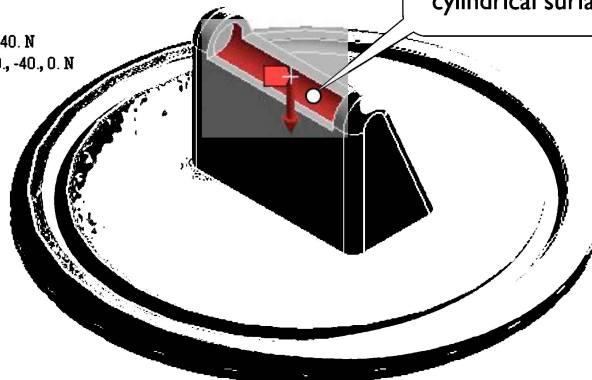


[4] Type -40 (N) for **Y Component**. #

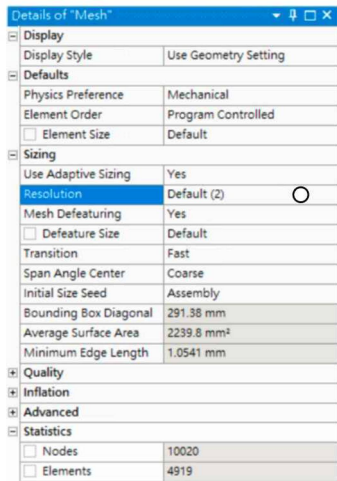
A: Static Structural
Bearing Load
Time: 1. s

Bearing Load: 40. N
Components: 0., -40., 0. N

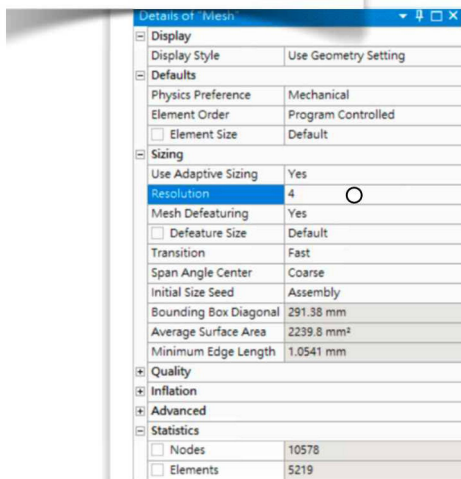
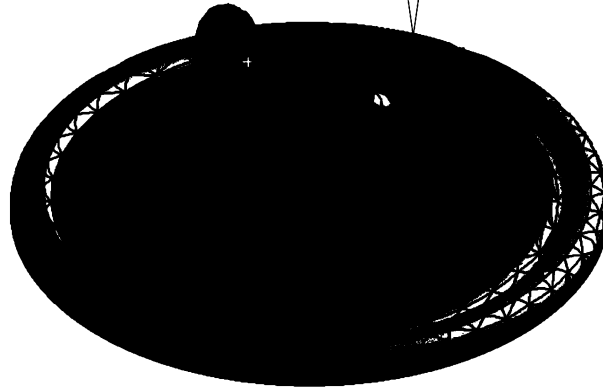
[3] The bearing load is applied on this cylindrical surface. ↑



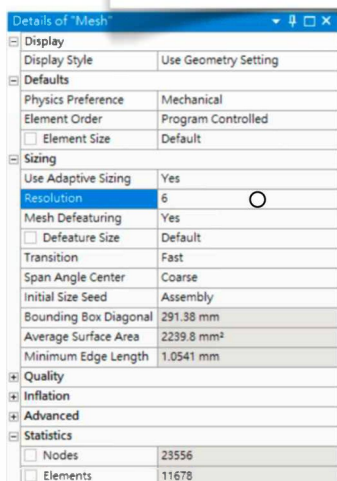
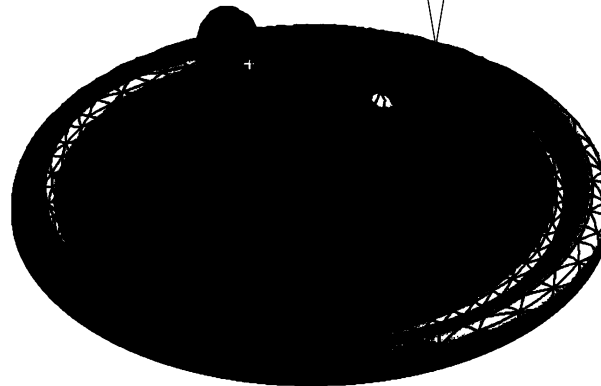
5.4.5 Generate Mesh



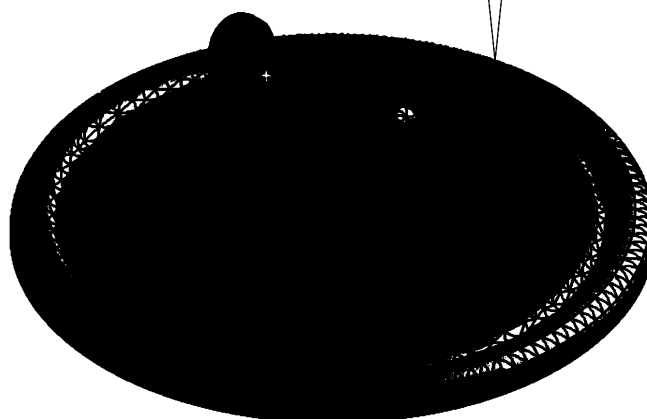
[1] The mesh generated with the default settings (**Resolution = 2**). ↓



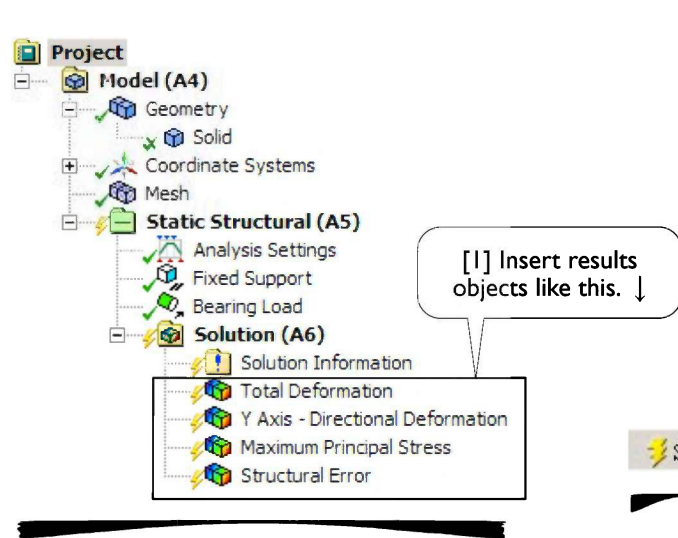
[2] The mesh generated with **Resolution = 4**. ↓





[3] The mesh generated with **Resolution = 6**. Let's use this mesh. #



5.4.6 Set Up Solution Branch and Solve



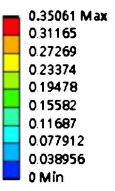
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Directional Deformation
Orientation	Y Axis 
By	Time
Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No

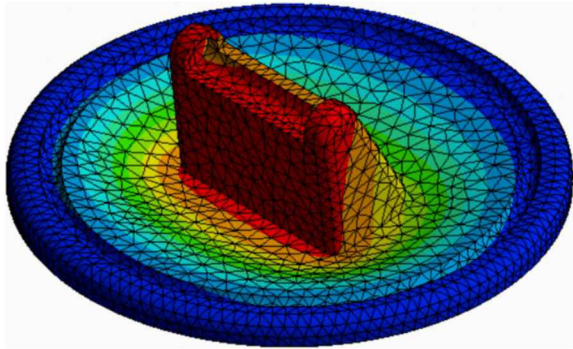


[2] Solve. #

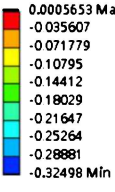
5.4.7 View the Results

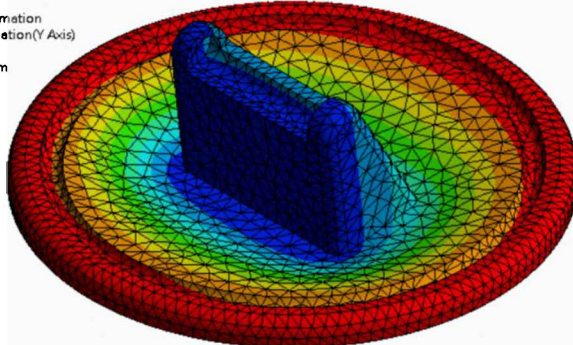
A: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1





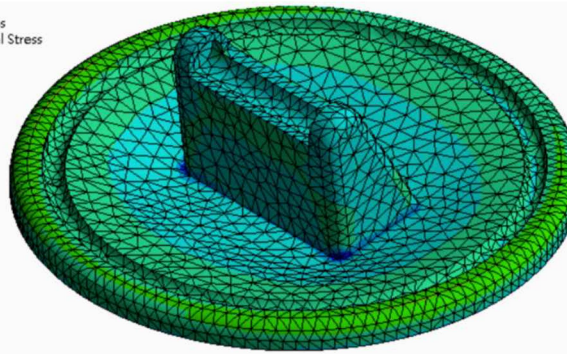
A: Static Structural
Y Axis - Directional Deformation
Type: Directional Deformation(Y Axis)
Unit: mm
Global Coordinate System
Time: 1





A: Static Structural
Maximum Principal Stress
Type: Maximum Principal Stress
Unit: MPa
Time: 1

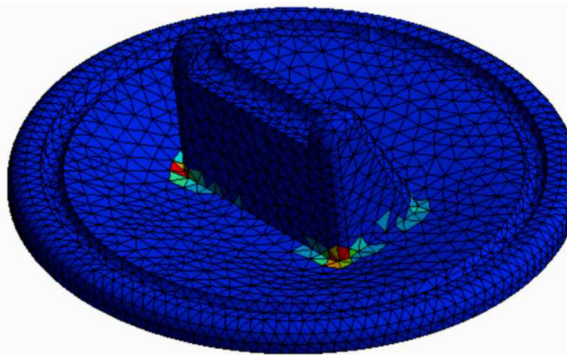
3.8774 Max
3.2564
2.6354
2.0143
1.3933
0.77224
0.15121
-0.46983
-1.0909
-1.7119 Min



[3] Maximum principal stress. ↓

A: Static Structural
Structural Error
Type: Structural Error
Unit: mJ
Time: 1

0.014707 Max
0.013073
0.011439
0.009805
0.0081708
0.0065367
0.0049025
0.0032683
0.0016342
1.4734e-8 Min



[4] Structural error. ↓

Wrap Up

[5] Close **Mechanical**, save the project, and exit Workbench. #

Section 5.5

Review

5.5.1 Keywords

Choose a letter for each keyword, from the list of descriptions

1. () Adaptive Meshing
2. () Bearing Load
3. () Coordinate System
4. () MultiZone Method
5. () Sweep Method

Answers:

1. (E) 2. (A) 3. (B) 4. (D) 5. (C)

List of Descriptions

(A) In 3D simulations, it applies on cylindrical faces. The total force is distributed on the compressive side of the cylindrical faces.

(B) To define a coordinate system, we need to specify the type of coordinate system, the location of origin, and the axes. The types of coordinate systems may be **Cartesian** or **Cylindrical** in the current version of the Workbench.

(C) A meshing method, in which the source faces, selected manually or automatically, are meshed, and the 2D mesh grows to become 3D elements by sweeping along a path up to target faces. Not all 3D solid bodies are sweepable. A sweepable 3D solid body often can be meshed with hexahedral elements.

(D) A meshing method. For a non-sweepable 3D solid body, the method tries to decompose the body into several sweepable bodies and then uses Sweep method to mesh each body.

(E) An automatic and iterative solution process to meet a user-specified solution accuracy. The basic idea is to refine the mesh size in the area of large structural errors until the specified accuracy is satisfied.

5.5.2 Questions

Symmetries

We didn't take any advantage of symmetries in all three simulation cases of this chapter. Point out the planes of symmetry in each model.

5.5.3 Additional Workbench Exercises

Simulation with Symmetric Models

It is a good exercise to redo the simulations in this chapter, taking advantage of the symmetries.

Additional Exercises

Additional Exercises of 3D simulations can be found in the Verification Manual for Workbench^[Ref 1].

Reference

1. All Help>Verification Manuals>ANSYS Workbench Verification Manual

Chapter 6

Surface Models

Many real-world objects can be modeled as surface bodies. For example, in Section 4.3, we modeled the glass as a surface body. The 2D bodies in Chapter 3, although created as surfaces, are called 2D solid bodies in **Mechanical**. Remember that 2D solid bodies do not have out-of-plane bending. This section focuses on the simulations of 3D surface bodies.

When a real-world body is thin enough, it is usually a good candidate for a 3D surface body. Workbench will mesh surface bodies with shell elements (1.3.3[9-11], page 38). There are many advantages of using surface models over 3D solid models. First, creating surface models is usually easier. Second, the problem size is much smaller; that implies a much smaller computing time. Third, it often results in a more accurate solution due to the efficiency of shell elements. Therefore, engineers should consider using surface models instead of solid models whenever possible. In the old days, surface models were visually awkward since they had zero thickness and occupied zero volume in the space. Workbench avoids this awkwardness by allowing a rendering of thickness, so that the surface bodies can be visually the same as solid bodies.

Purpose of This Chapter

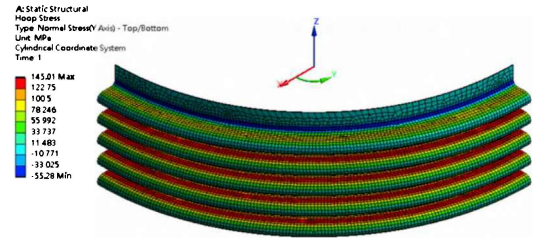
The main purpose of this chapter is to introduce the use of shell elements. This chapter guides the students to create surface models and perform simulations using surface models. The chapter uses three examples; two of them are purely surface models while the other one is a model mixed up with solid bodies and surface bodies.

About Each Section

Section 6.1 creates a bellows joint model and performs a simulation. In Section 6.2, the bracket, which we introduced in Sections 4.1 and 5.1, is used again to demonstrate how an existing solid model can be transformed to a surface model using the tool **Mid-Surface**. The solution is numerically comparable with that obtained using the solid model in Section 5.1, while using much less computing time. Section 6.3 demonstrates the simulation of a model that has a mix-up of solid bodies and surface bodies.

Section 6.1

Bellows Joints



6.1.1 About Bellows Joints

[1] The bellows joints [2-3] are used to absorb thermal or vibrational movement in a pipeline system that transports high pressure gases; they are designed to sustain internal pressure as well as external pressure. The external pressure is considered when the piping system is used under the ocean. With the internal pressure, the engineers are concerned about its radial deformation (due to a tolerance consideration) and hoop stress (due to a safety consideration). With the external pressure, buckling is the major concern, which will be discussed in 10.4.2, page 388. ↓

[4] In this section, we will perform a static structural simulation under the internal pressure of 0.5 MPa.

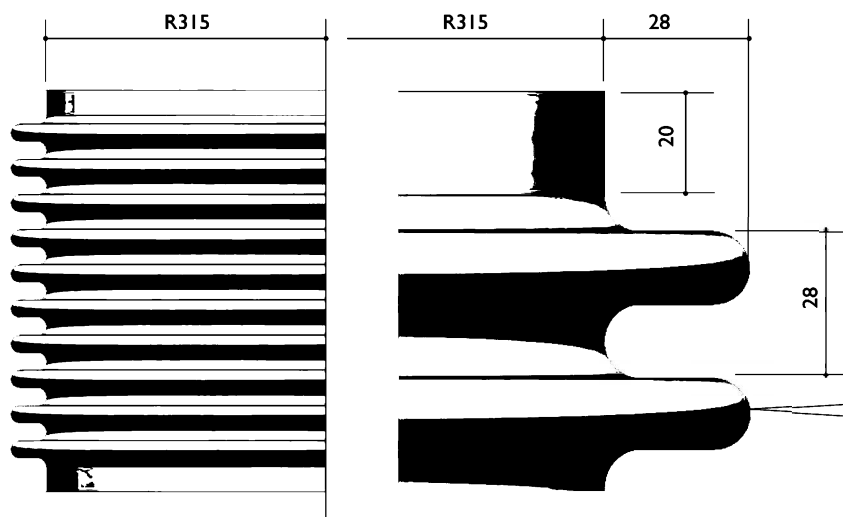
The bellows joint is axisymmetric both in geometry and loading. It is also symmetric about a horizontal plane. We will take advantage of these symmetries and model 1/8 of the bellows joint as a 3D surface body.

We might model the bellows joint as an axisymmetric 2D solid body; however, it would result in a poorer solution than 3D surface body for this particular case, in which the bending dominates the structural behavior. #



[2] The bellows joints are made of an SU316 steel, which has a Young's modulus of 180 GPa and a Poisson's ratio of 0.28. ↓

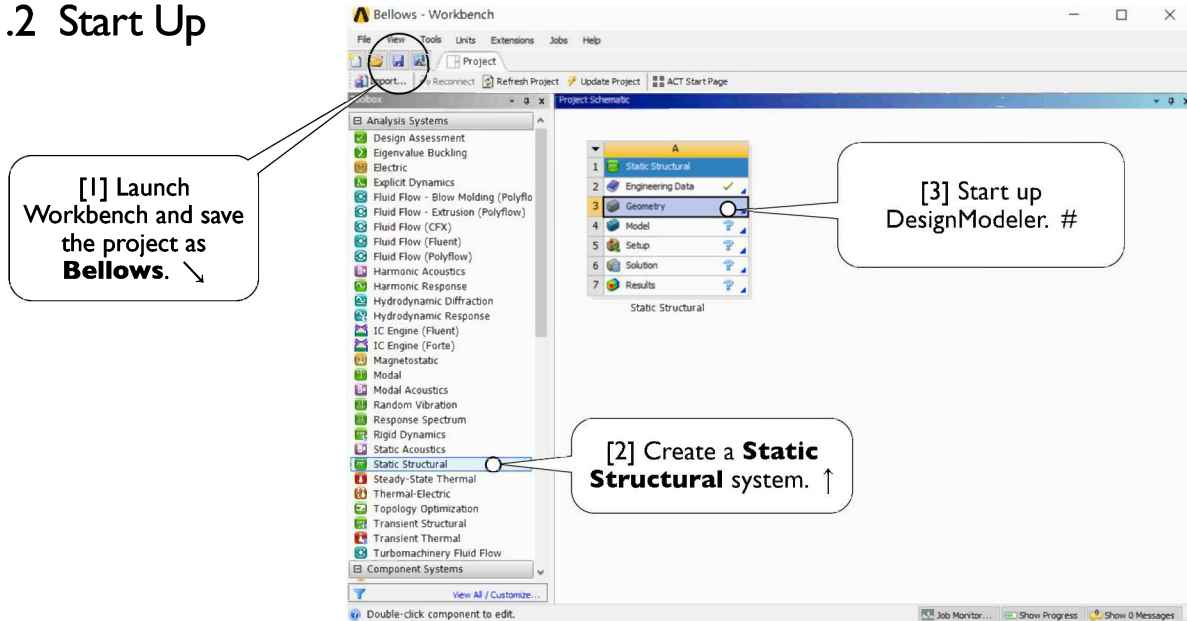
Unit: mm.



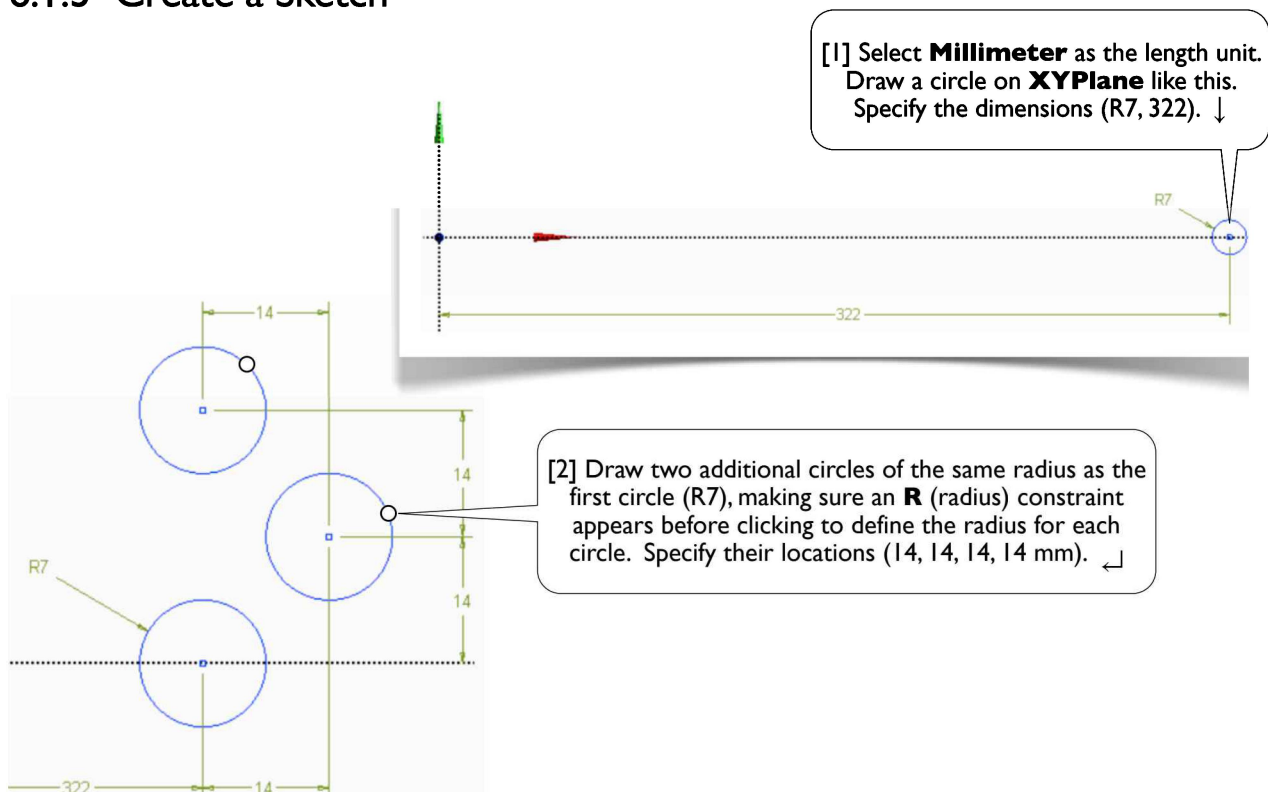
[3] All the arcs have radii of 7 mm. The thickness of the sheet steel is 0.8 mm. ↗

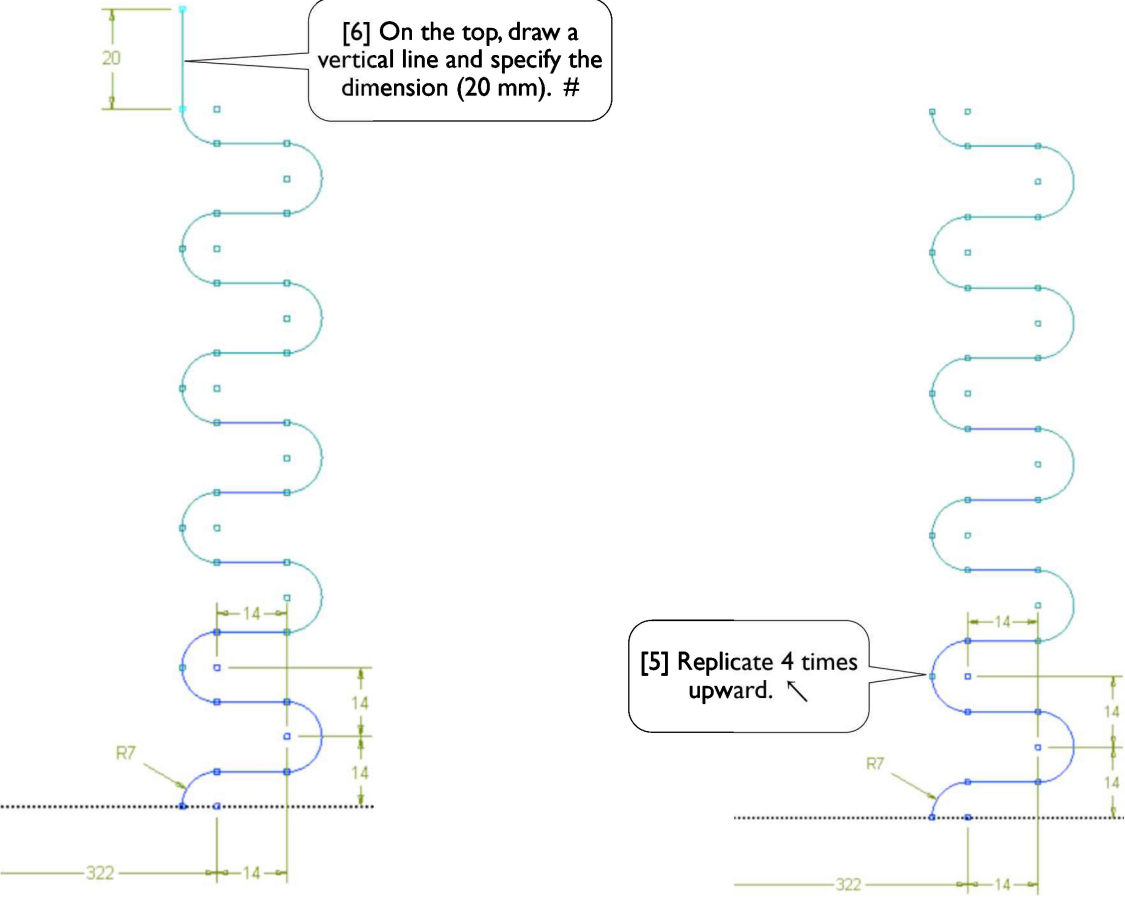
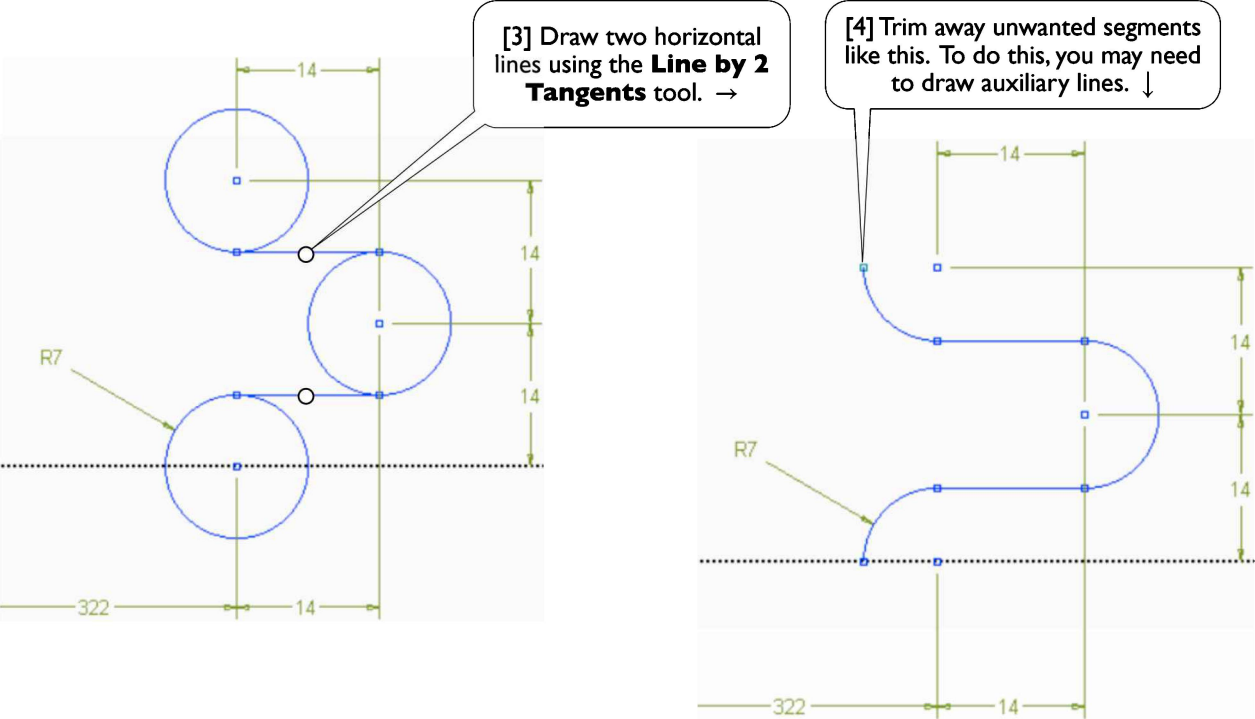
PART A. GEOMETRIC MODELING

6.1.2 Start Up



6.1.3 Create a Sketch





6.1.4 Create Surface Body

[1] Click **Revolve** on the toolbar. →

[2] Click to bring forth **Apply/Cancel** buttons and select the Y-axis in the graphics window. ↓

[3] Type 90 (degrees). ←

[4] Click **Generate**. ↘

[5] A 3D surface body is created. Rotate to view the model isometrically. #

6.1.5 Specify Symmetry Conditions

[1] Pull-down-select **Tools/Symmetry** (3.1.11[7], page 119). Select 3 for **Number of Planes**. ↓

[2] From the model tree, select **XYPlane** for **Symmetry Plane 1**, **ZXPlane** for **Symmetry Plane 2**, and **YZPlane** for **Symmetry Plane 3**, respectively. →

[3] Click **Generate**. The symmetry conditions are imposed, although you don't see any changes of the model. These boundary conditions will be exported to **Mechanical**. Close DesignModeler. #

PART B. SIMULATION

6.1.6 Prepare Material Properties

[1] Double-click **Engineering Data** to add a material. ✓

[2] Type **SU316** to add a new material to **Engineering Data**. ✓

[3] Double-click **Isotropic Elasticity** to include the material properties. →

[4] Type **1.8E+11 (Pa)** for **Young's Modulus** and **0.28** for **Poisson's Ratio**. ↖

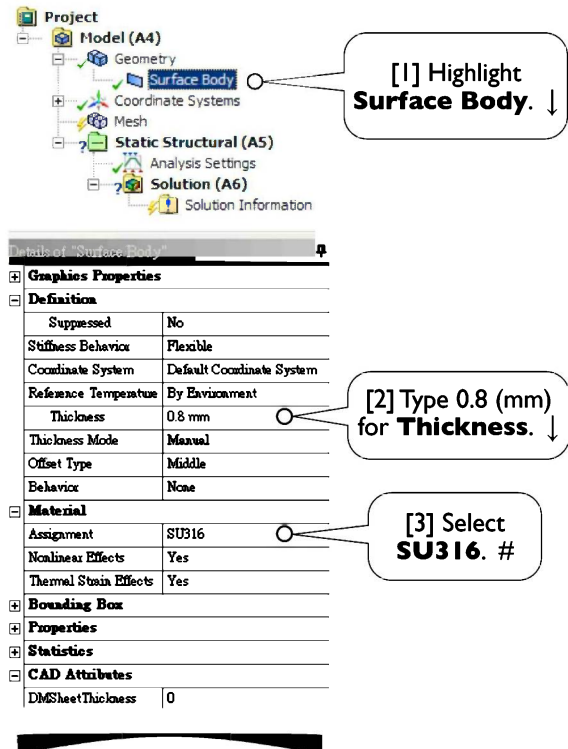
[5] Click to return to **Project Schematic**. ↘

[6] Start up **Mechanical**. And make sure the unit system is **mm-kg-N-s. #**

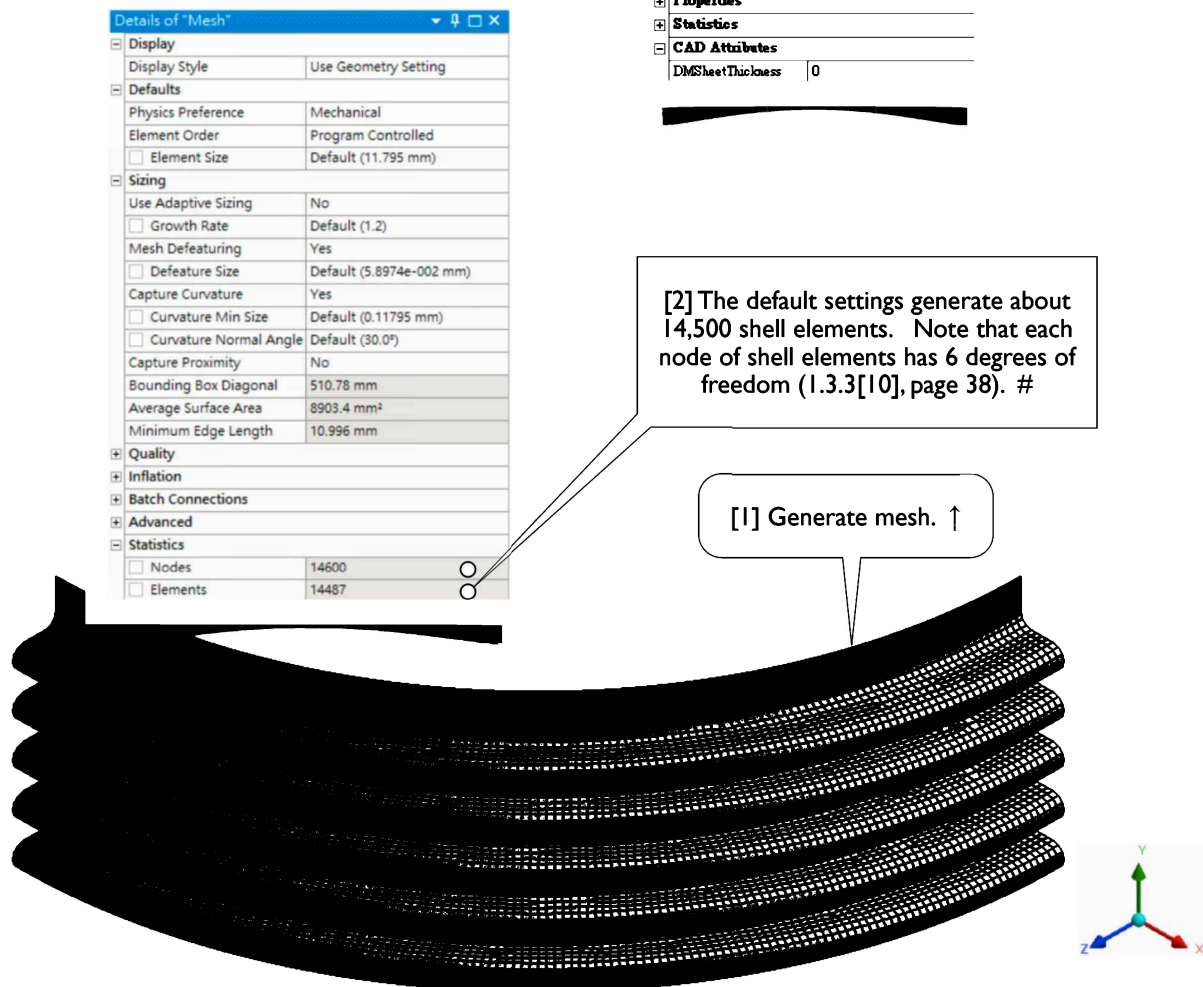
	A	B	C	D
1	Contents of Engineering Data	source		Description
2	Material			
3	Structural Steel			Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
4	SU316			
*	Click here to add a new material			

	A	B	C	D	E
1	Property	Value	Unit		
2	Isotropic Elasticity				
3	Derive from	Young's Modulus and Poisson's Ratio			
4	Young's Modulus	1.8E+11	Pa		
5	Poisson's Ratio	0.28			
6	Bulk Modulus	1.3636E+11	Pa		
7	Shear Modulus	7.0313E+10	Pa		

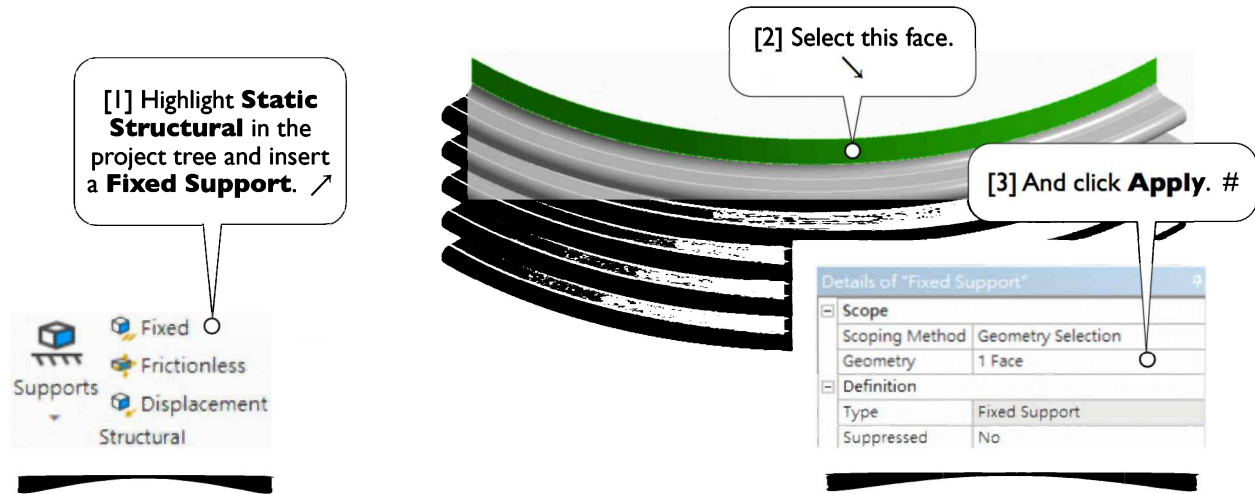
6.1.7 Assign Material and Thickness



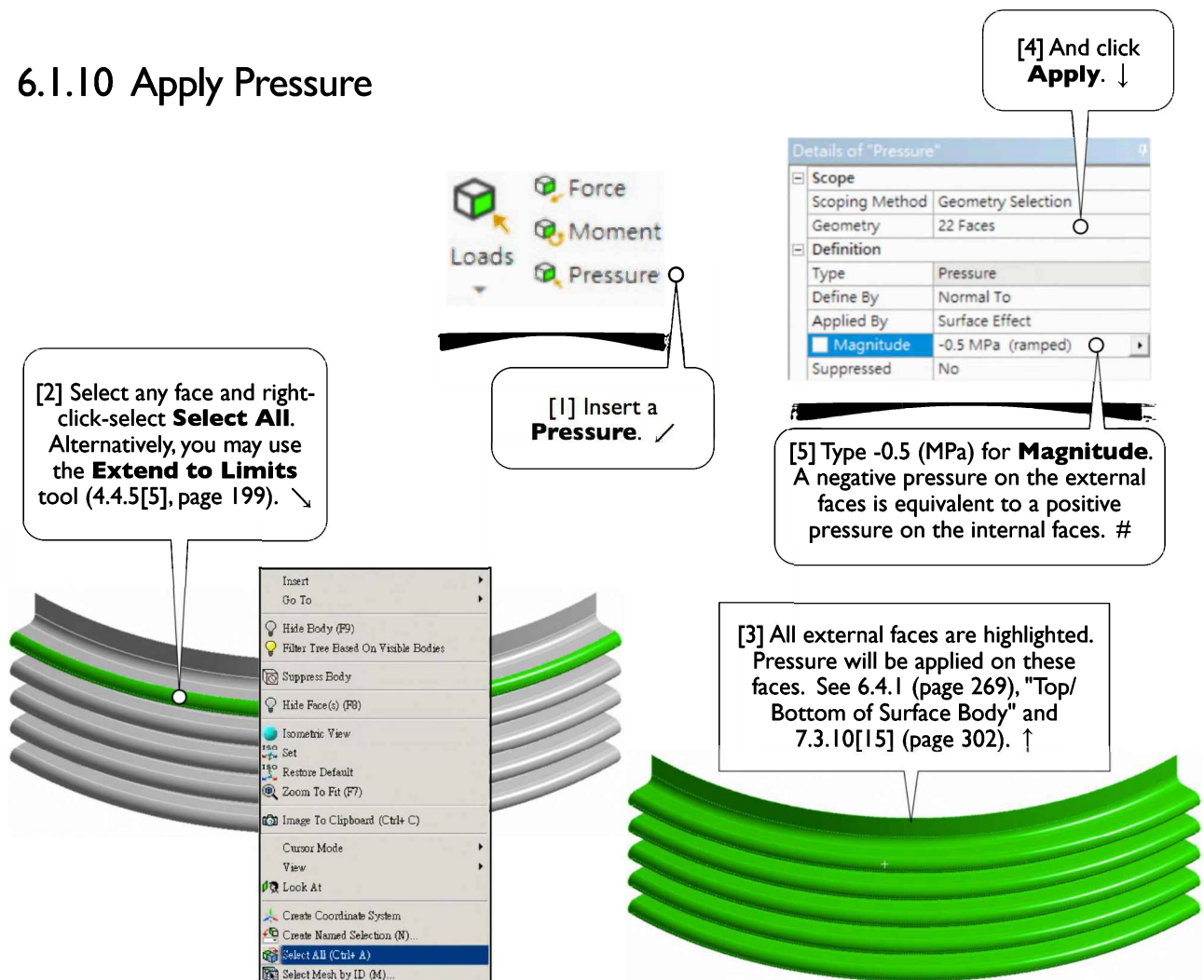
6.1.8 Generate Mesh



6.1.9 Specify Supports



6.1.10 Apply Pressure



6.1.11 Create a Cylindrical Coordinate System

Details of "Coordinate System"

Definition	
Type	Cylindrical
Coordinate System	Program Controlled
APDL Name	
Suppressed	No
Origin	
Define By	Global Coordinates
Origin X	0. mm
Origin Y	0. mm
Origin Z	0. mm
Location	Click to Change
Principal Axis	
Axis	Z
Define By	Global Y Axis
Orientation About Principal Axis	
Axis	X
Define By	Global Z Axis
Directional Vectors	
Transformations	
Base Configuration	Absolute
Transformed Configuration	[0. 0. 0.]

[1] Highlight **Coordinate Systems** in the project tree, and click **Coordinate System** in the toolbars (5.2.10[2], page 224). ←

[2] Set up the coordinate system like this. →

[3] The cylindrical coordinate system should look like this. ↓

[4] In the project tree, rename the newly created coordinate system to **Cylindrical Coordinate System**. #

6.1.12 Set Up Solution Branch and Solve the Model

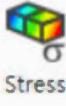
[1] Highlight **Solution** in the model tree, and insert a **Deformation/Directional**. →

Details of "Directional Deformation"

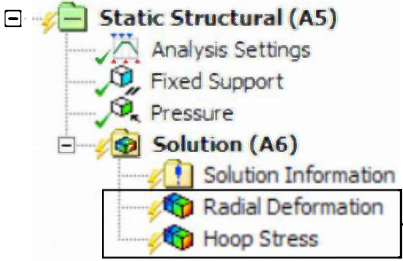
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Directional Deformation
Orientation	X Axis
By	Time
Display Time	Last
Coordinate System	Cylindrical Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
Results	
Minimum	
Maximum	
Minimum Occurs On	
Maximum Occurs On	
Information	

[3] Since a cylindrical coordinate system is used, the X-direction should be read "radial" direction. ←

[2] In the **Details**, select **Cylindrical Coordinate System**. ↑



[4] Insert a **Stress/Normal**. →



Details of "Normal Stress"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Position	Top/Bottom
Definition	
Type	Normal Stress
Orientation	Y Axis
By	Time
Display Time	Last
Coordinate System	Cylindrical Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
Results	
Minimum	
Maximum	
Minimum Occurs On	
Maximum Occurs On	
Information	

[6] Select **Y Axis** for **Orientation**. The Y-direction should be read "hoop" direction. ✓

[5] Select **Cylindrical Coordinate System**. ↑

[7] Rename for better readability. →

[8] Click **Solve**. #

6.1.13 View the Results

1.0 (True Scale)

Scoped Bodies

☒ Large Vertex Contours

[1] Select **Radial Deformation** in the project tree. The default scale is too exaggerated; select **True Scale**. ←

[2] The maximum radial deformation. ↙

A: Static Structural
Radial Deformation
Type: Directional Deformation(X Axis)
Unit: mm
Cylindrical Coordinate System
Time: 1

0.16687 Max

0.14833

0.12979

0.11125

0.092708

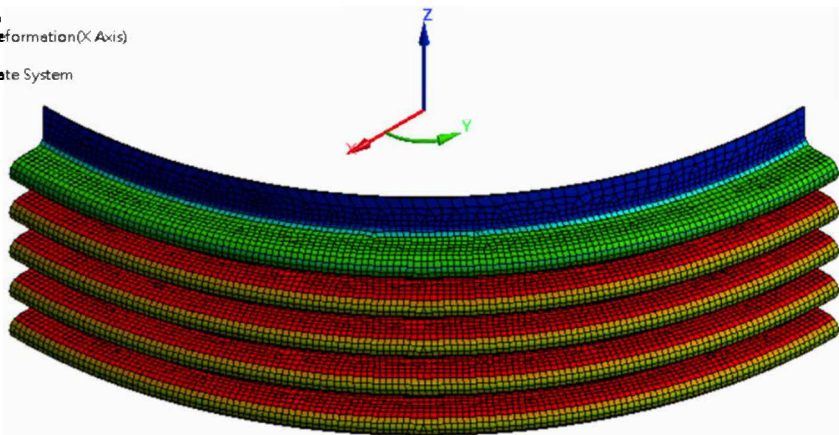
0.074166

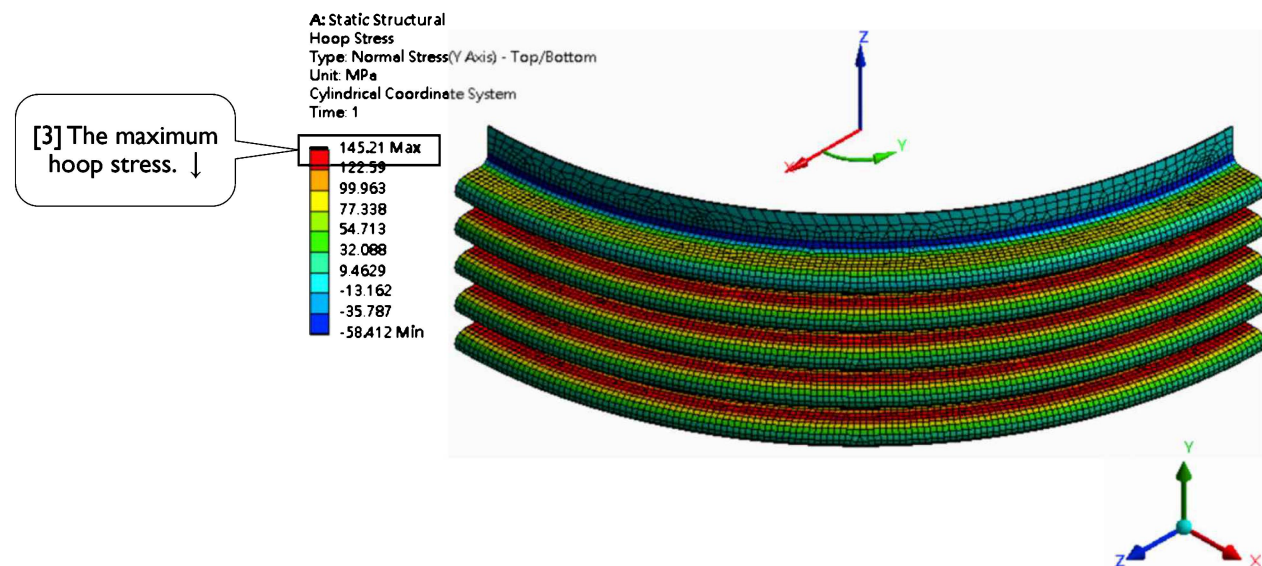
0.055625

0.037083

0.018542

0 Min





Wrap Up

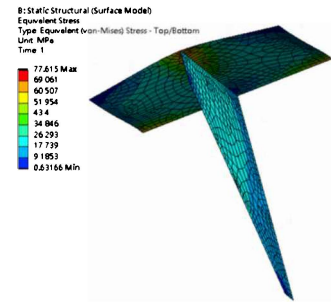
[4] Save the project and exit Workbench. #

Reference

1. All Help>Mechanical APDL>Theory Reference>13.208. SHELL208

Section 6.2

Beam Bracket



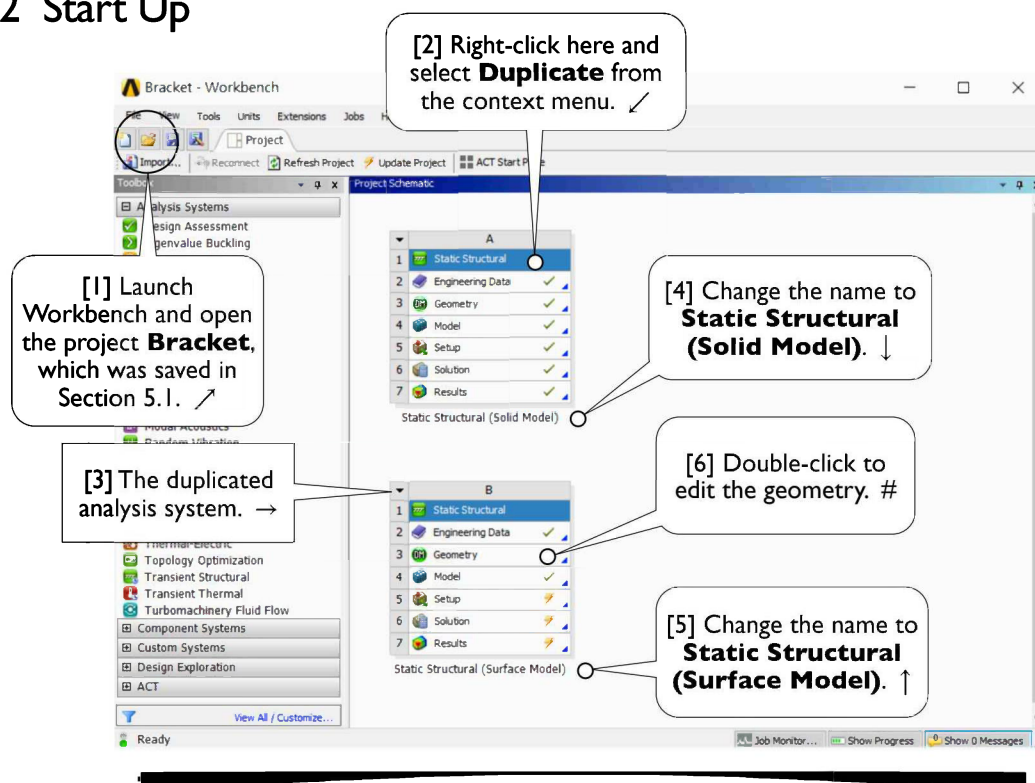
6.2.1 About the Beam Bracket

[1] In Section 4.1, we created a 3D solid model for the beam bracket, and the model was simulated in Section 5.1. Since the seat plate (flange) and the web plate are relatively thin and have uniform thicknesses, is it possible to model the beam bracket as a surface model and obtain a comparable result?

To create a surface model for the beam bracket, we don't have to start from scratch; we can use a tool in DesignModeler, called **Mid-Surface**. In fact, surface models are often created this way. CAD models are usually created as 3D solids, since they are created for multiple purposes, and simulation is only one of them. When a surface model is needed for simulation, **Mid-Surface** is a powerful tool to extract a surface model from a solid model. #

PART A. GEOMETRIC MODELING

6.2.2 Start Up



6.2.3 Automatically Find Face Pairs

[1] Pull-down-select **Tools/Mid-Surface**. →

Details of MidSurf1	
Mid-Surface	MidSurf1
Face Pairs	0
Selection Method	Automatic
Bodies To Search	Visible Bodies
Minimum Threshold	10 mm
Maximum Threshold	16 mm
Find Face Pairs Now	No
FD3, Selection Tolerance (≥ 0)	No
FD1, Thickness Tolerance (≥ 0)	0.0005 mm
FD2, Sewing Tolerance (≥ 0)	0.02 mm
Extra Trimming	Intersect Untrimmed with Body
Preserve Bodies?	No

[2] Select **Automatic** to let the program find face pairs. Type 10 (mm) for **Minimum Threshold** and 16 (mm) for **Maximum Threshold**. ↓

[3] Select **Yes**. As soon as you select **Yes**, three face pairs are automatically found and displayed in the yellow area (also see [4]). ✓

[4] Three face pairs are found and colored. ✓

[5] Click **Generate**. ↓

[6] Two surface bodies are generated; they form a single part (see [7]). ↓

[8] This example is so simple that DesignModeler can automatically find all the face pairs. For complicated cases, you may need to manually select the face pairs. To illustrate manual selection, let's delete what we've done on this page, and redo it manually on the next page. Now, right-click **MidSurf1** and select **Delete (Suppress)** has the same effect). #

[7] The two surface bodies form a single part; i.e., they are treated as an integrated part. ↑

6.2.4 Manually Select Face Pairs

[1] Again, pull-down-select **Tools/Mid-Surface**. →

[2] Select this face. →

[3] Control-select this face. This and the last face form a *face pair*. **Mid-Surface** tool will extract the mid-surface of a face pair. ✓

[4] Select (without holding the Control key) this face. →

[5] Control-select this face. ✓

[6] Select (without holding the Control key) this face. ✓

[7] Control-select this face again. ✓

[8] Click **Apply** and select **Manual**. →

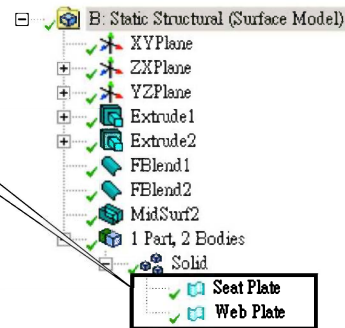
[9] Click **Generate**. ↓

[10] Two surface bodies are generated. Again, they form a single part (6.2.3[7], last page). ↵

Details View

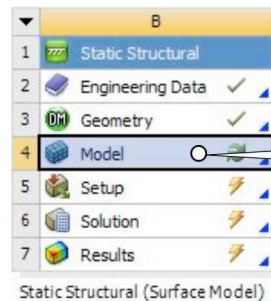
Details of MidSurf2	
Mid-Surface	MidSurf2
Face Pairs	3
Selection Method	Manual
FD3, Selection Tolerance (>=0)	0 mm
FD1, Thickness Tolerance (>=0)	0.0005 mm
FD2, Sewing Tolerance (>=0)	0.02 mm
Extra Trimming	Intersect Untrimmed with Body
Preserve Bodies?	No

[1] Rename the two surface bodies for better readability. Close DesignModeler. #



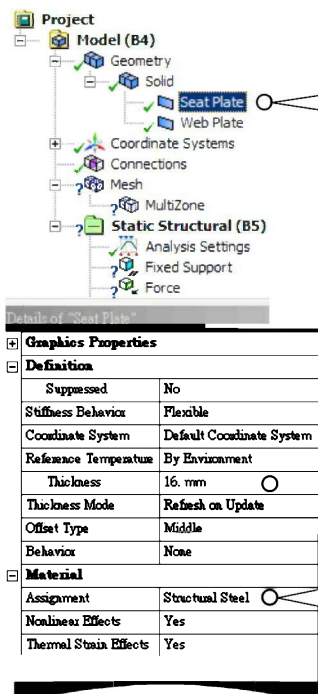
PART B. SIMULATION

6.2.5 Start Up **Mechanical**



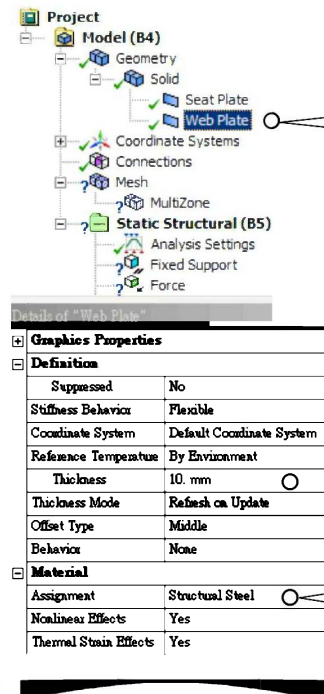
[1] Double-click to start up **Mechanical**. Click **Yes** to update the model. In **Mechanical**, make sure the unit system is **mm-kg-N-s**. #

6.2.6 Check Material and Thickness



[1] Highlight **Seat Plate**. ↓

[2] Make sure the thickness is 16 (mm) and select **Structural Steel** for **Assignment**. ↗



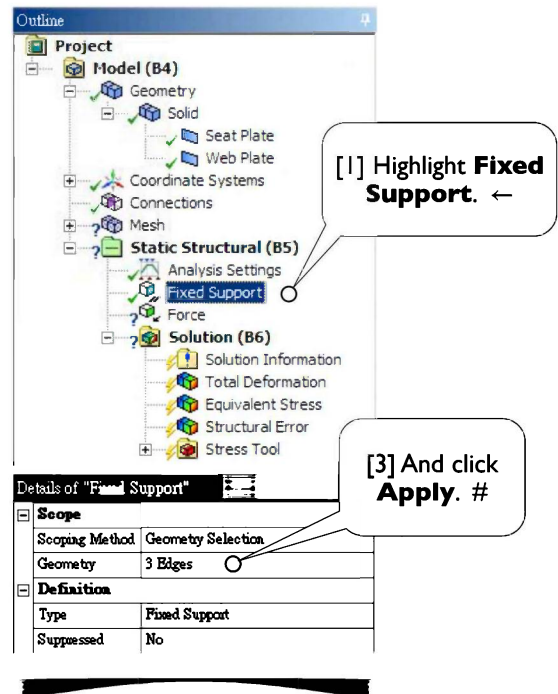
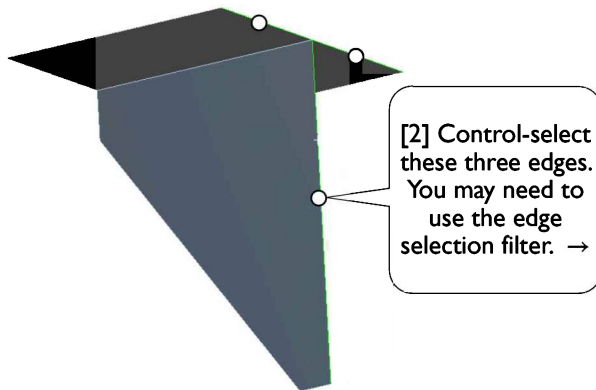
[3] Highlight **Web Plate**. ↓

[4] Make sure the thickness is 10 (mm) and select **Structural Steel** for **Assignment**. ↖

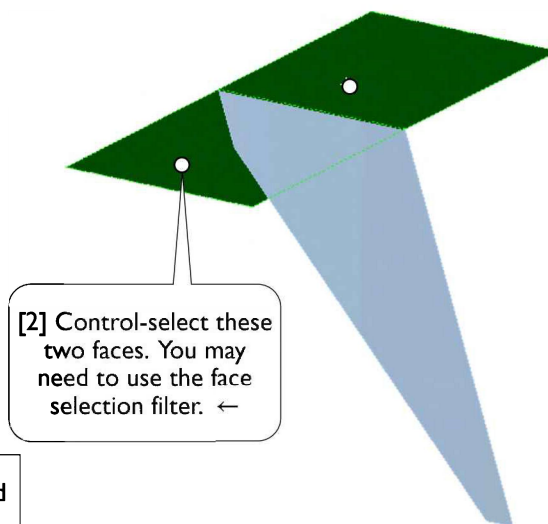
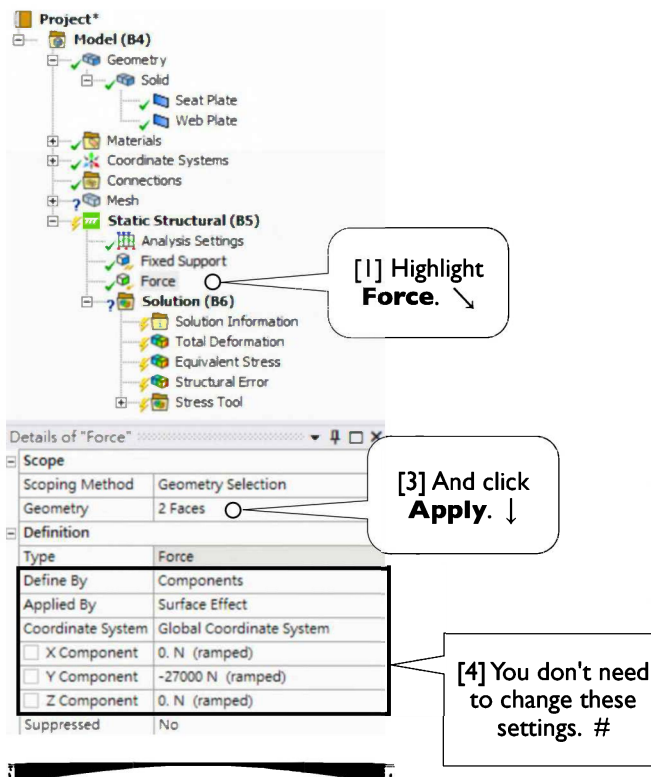
Thickness Assignment

[5] The surface bodies are created using the **Mid-Surface** tool. In most cases the thicknesses of bodies are correctly calculated and transferred to **Mechanical**. For some complicated cases, the thicknesses may not be correctly calculated and transferred. It is good practice to always check material and thickness assignments in **Mechanical**. #

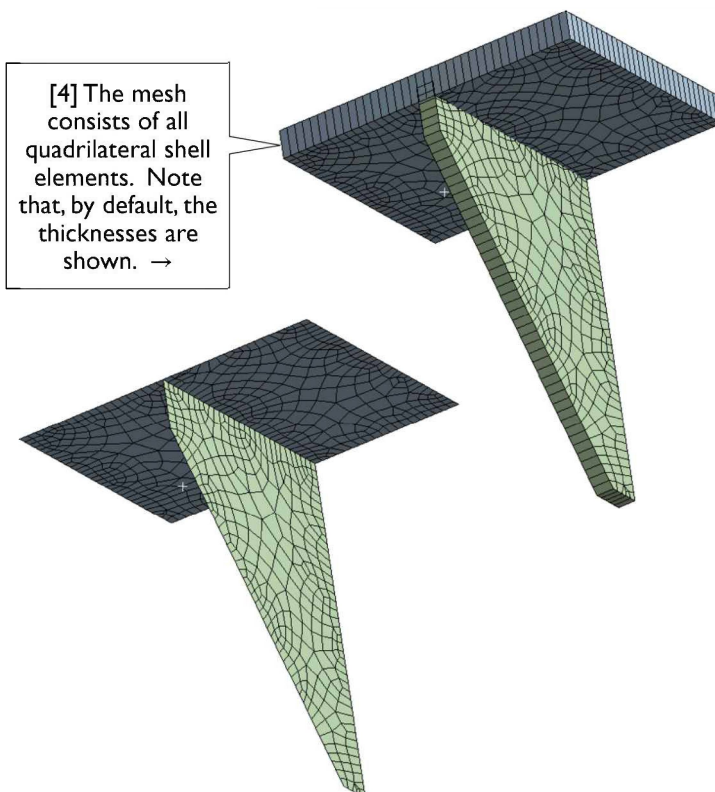
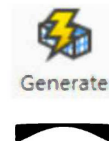
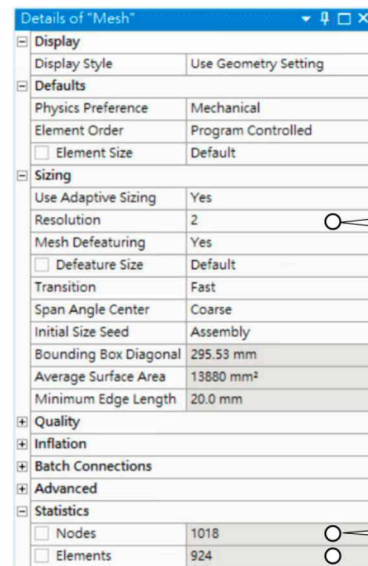
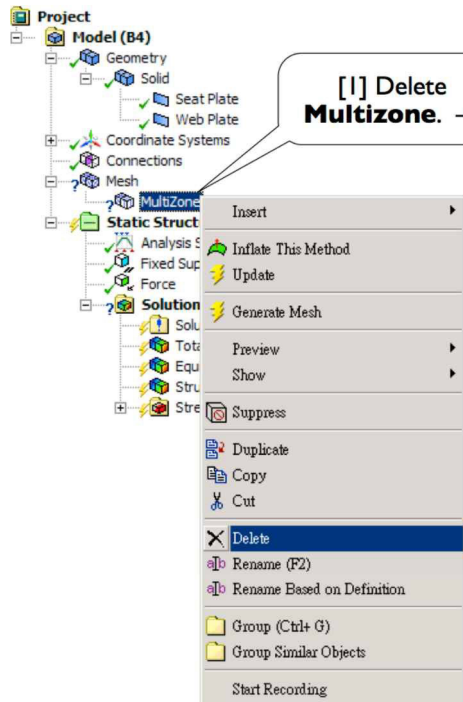
6.2.7 Redefine Fixed Support



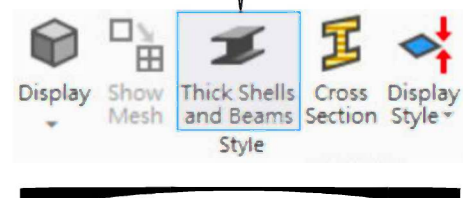
6.2.8 Redefine Force



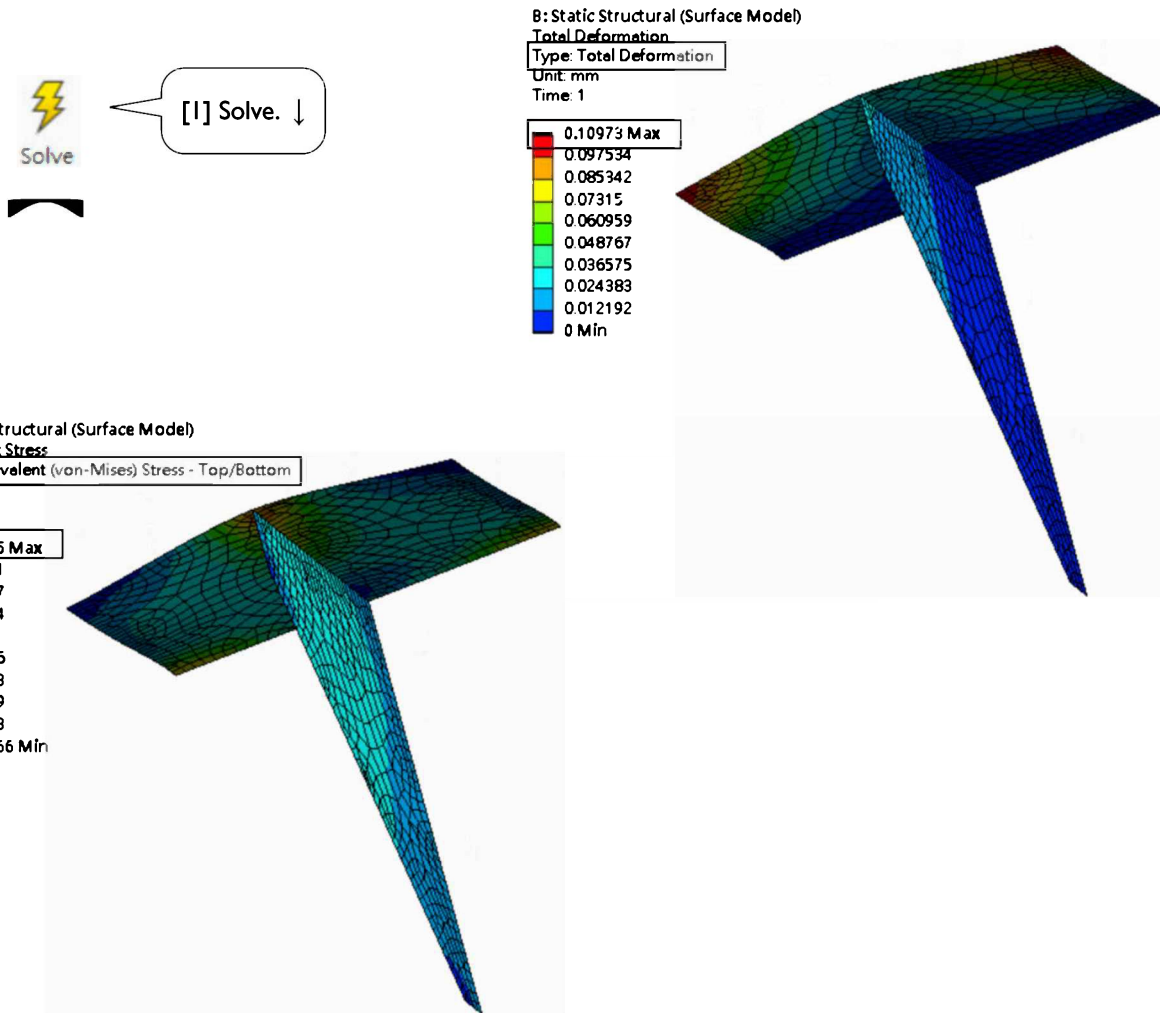
6.2.9 Generate Mesh



[5] You may turn off the thicknesses display by de-selecting **Display/Thick Shells and Beams** in the toolbars. #



6.2.10 Solve the Model



Surface Model versus Solid Model

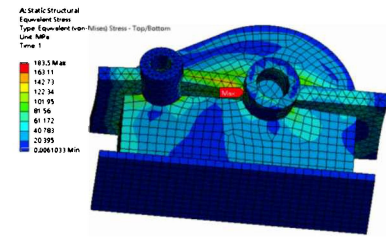
[2] The results are quite comparable with those obtained in 5.1.13[8-9] (page 218). The surface model in this section consists of 6,108 degrees of freedom ($6 \text{ DOFs} \times 1,018 \text{ nodes} = 6108 \text{ DOFs}$, see 6.2.9[3], last page) while the solid model consists of 27,651 degrees of freedom ($3 \text{ DOFs} \times 9217 = 27,651 \text{ DOFs}$, see page 217)! In general, before you decide to use a solid model, reconsider the possibility of using a surface (or line) model. ↓

Wrap Up

[3] Save the project and exit Workbench. #

Section 6.3

Gearbox



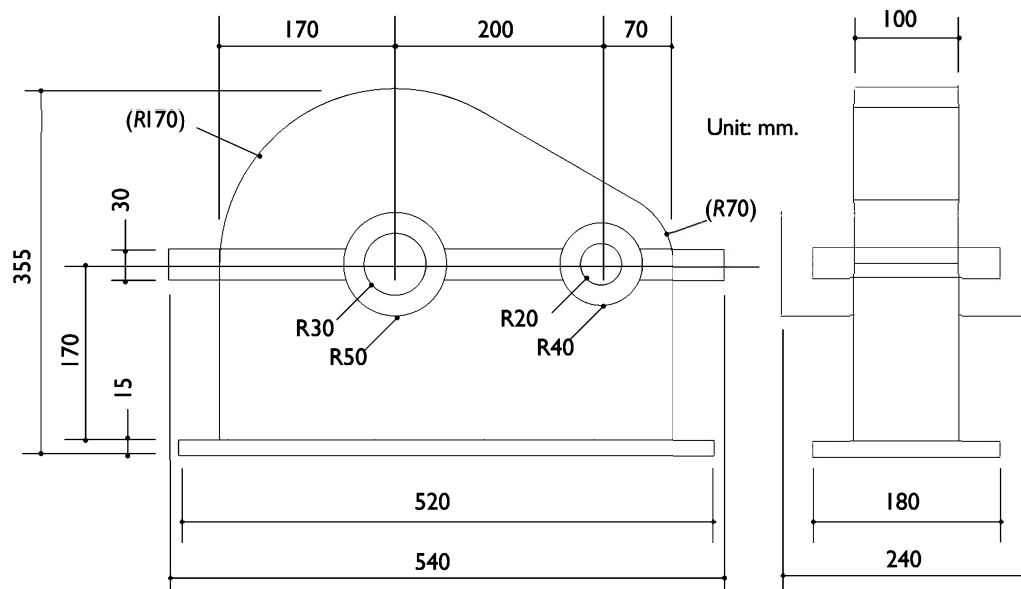
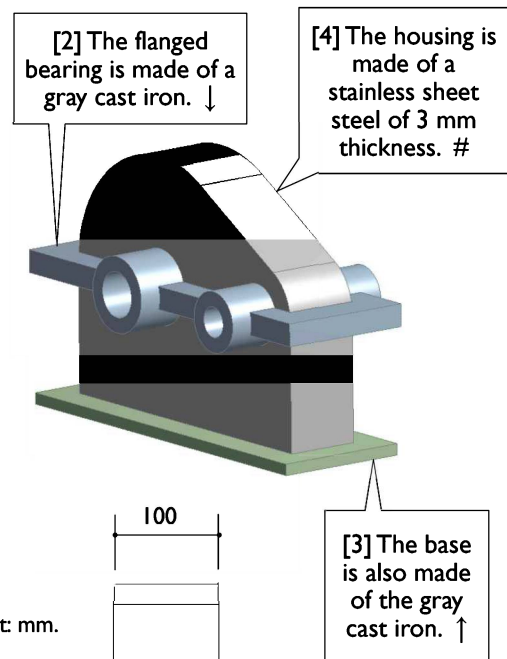
In DesignModeler, a 3D surface body is usually created using one of four techniques: (a) Creating a 2D sketch and applying **Concept/Surface From Sketches** to generate a planar surface body; this technique has been demonstrated in Section 4.3 to create the glass. (b) Creating an open sketch and then using **Extrude, Revolve, Sweep, or Skin/Loft** to generate planar or non-planar surfaces; this technique has been demonstrated in Section 6.1 to create the bellows. (c) Applying **Tool/Mid-Surface** to extract mid-surface from a solid body; this technique has been demonstrated in Section 6.2 to create a surface model for the beam bracket. (d) Using **Thin/Surface** tool; the idea is to extract the exterior "shell" of a solid body. We will demonstrate this technique in this section.

6.3.1 About the Gearbox^[Ref 1]

[1] Gears are used to transmit mechanical power. During the power transmission, the gears exert forces on each other. Through the gear shafts, the forces eventually reach the bearing supports. A gearbox is designed to withstand the bearing forces.

In this case, engineers are concerned about the deformation of the gearbox, for it may cause a displacement of the rotating axes, which may reach a point where the transmission becomes defective.

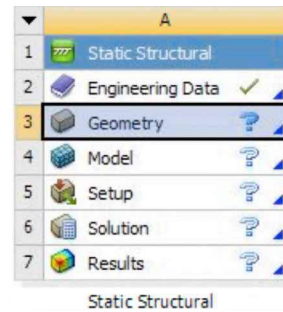
In this section, we will create a 3D model for the gearbox and conduct a simulation. The model will consist of two solid bodies for the flange and the base, made of a gray cast iron [2-3], and a surface body for the housing, made of a stainless steel [4]. →



PART A. GEOMETRIC MODELING

6.3.2 Start Up

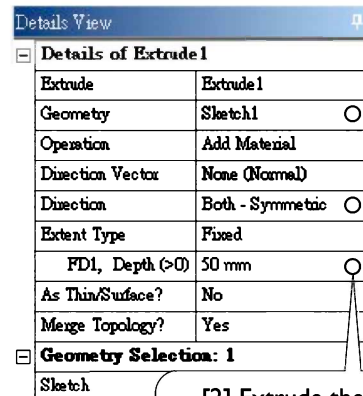
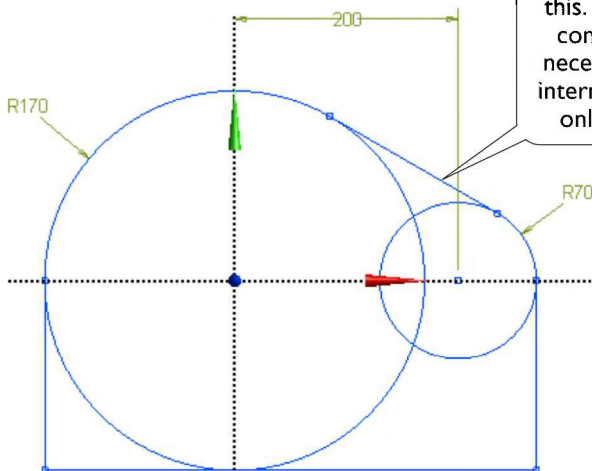
[1] Launch Workbench. Create a **Static Structural** system. Save the project as **Gearbox**. →



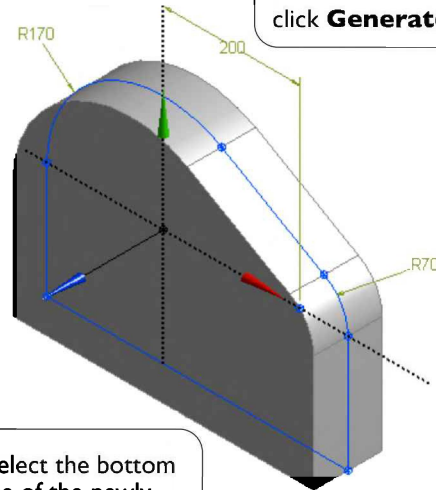
[2] Start up DesignModeler. #

6.3.3 Create the Housing

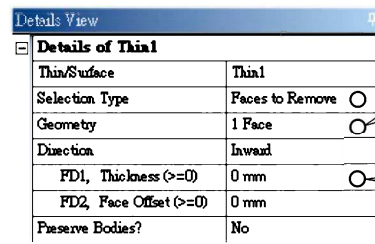
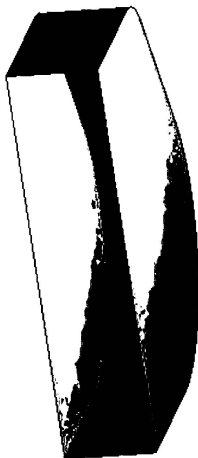
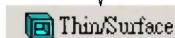
[1] Select **Millimeter** as the length unit. Create a sketch on **XYPlane** like this. Impose **Tangent** constraints wherever necessary. Trim away all internal segments. Leave only the outlines. →



[2] Extrude the sketch 50 mm both sides. Remember to click **Generate**. ←



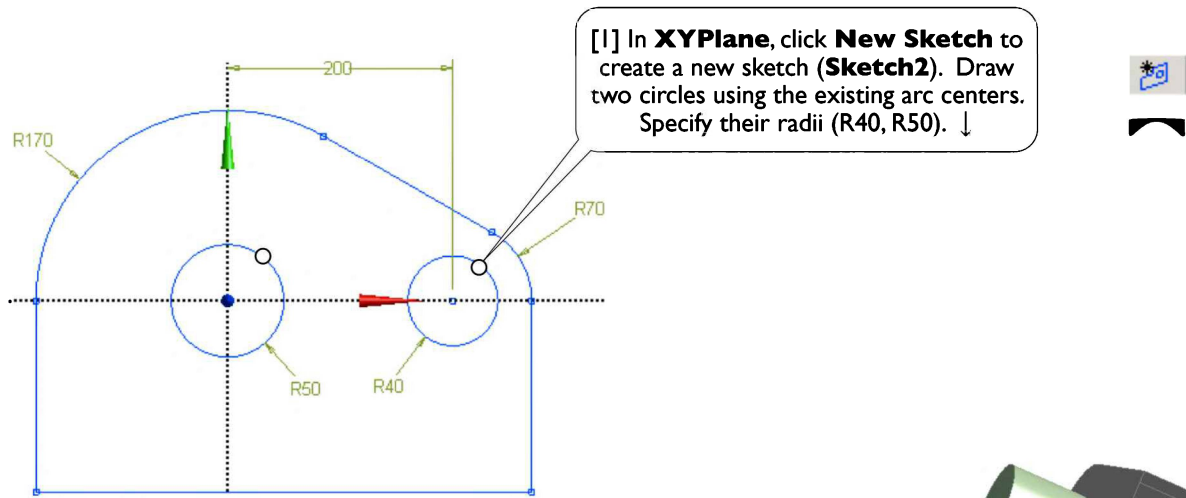
[3] Click **Thin/Surface** on the toolbar. ↓



[4] Select the bottom face of the newly created solid body. ↓

[5] To create a surface body, you must type a zero value for the thickness. If you type a nonzero value, it would create a "thin solid body" (see 4.5.5[1-2], page 206) instead of a surface body. Remember to click **Generate**. #

6.3.4 Create the Bearings



Details of Extrude2	
Extrude	Extrude2
Geometry	Sketch2
Operation	Add Material
Direction Vector	None (Normal)
Direction	Both - Symmetric
Extent Type	Fixed
FD1, Depth (>0)	120 mm
As Thin/Surface?	No
Merge Topology?	Yes
Geometry Selection: 1	
Sketch	Sketch2

[2] Extrude **Sketch2** 120 mm both sides. Remember to click **Generate**. ↓

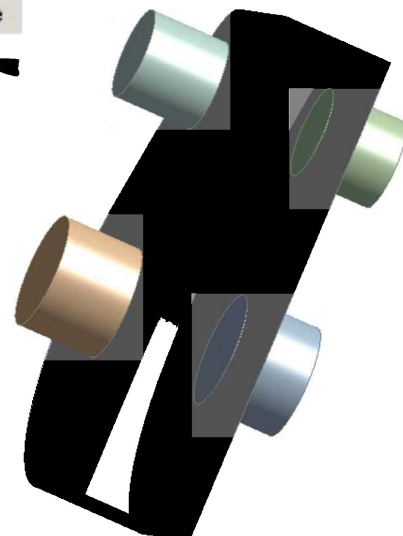
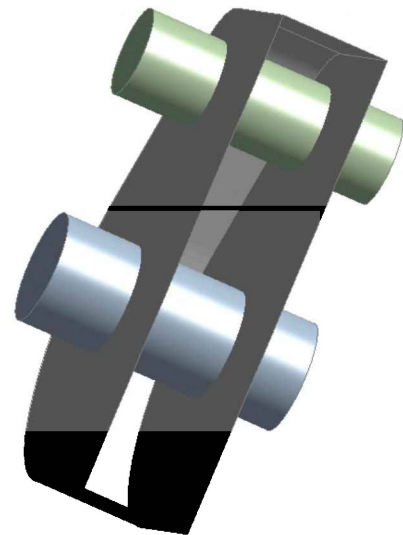
[3] Click **Extrude** again. ↓

Details of Extrude3	
Extrude	Extrude3
Geometry	Sketch2
Operation	Cut Material
Direction Vector	None (Normal)
Direction	Both - Symmetric
Extent Type	Fixed
FD1, Depth (>0)	50 mm
As Thin/Surface?	No
Target Bodies	All Bodies
Merge Topology?	Yes
Geometry Selection: 1	
Sketch	Sketch2

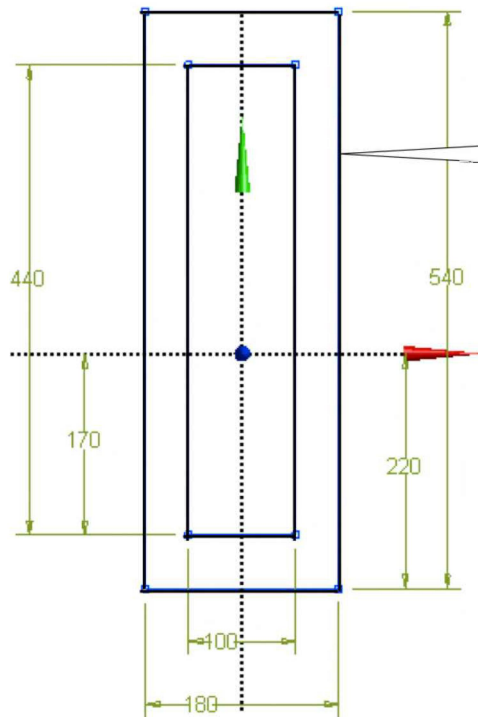
[4] Click **Apply**. **Sketch2** (current sketch) is used again. ↓

[5] Select **Cut Material**. ↓

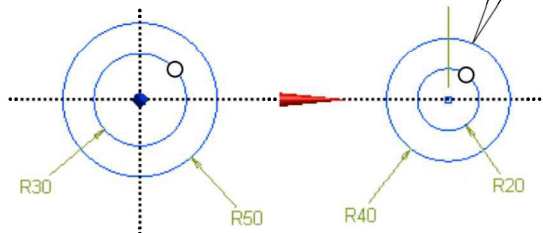
[6] Cut 50 mm for both sides. Remember to click **Generate**. #



6.3.5 Create the Flange



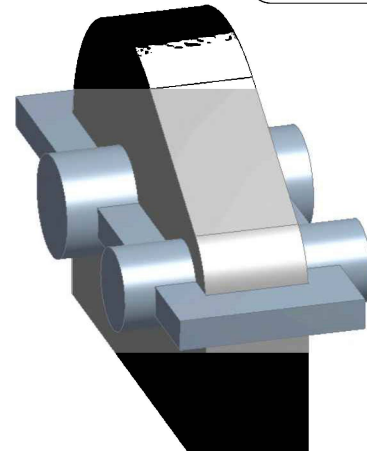
[1] In **ZXPlane**, draw a sketch like this. Remember to impose **Symmetry** constraints (about the vertical axis). →



[3] In **XYPlane**, click **New Sketch** to create a new sketch (**Sketch4**). Draw two smaller circles (than existing ones). Specify their radii (R20, R30). ↓

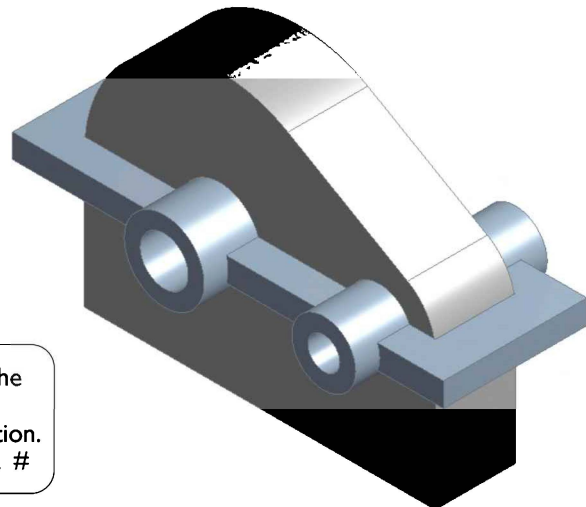
Details of Extrude4	
Extrude	Extrude4
Geometry	Sketch3
Operation	Add Material
Direction Vector	None (Normal)
Direction	Both - Symmetric
Extent Type	Fixed
FD1, Depth (>0)	15 mm
As Thin/Surface?	No
Merge Topology?	Yes
Geometry Selection: 1	
Sketch	Sketch3

[2] Extrude 15 mm both sides. Click **Generate**. ←



Details of Extrude5	
Extrude	Extrude5
Geometry	Sketch4
Operation	Cut Material
Direction Vector	None (Normal)
Direction	Both - Symmetric
Extent Type	Through All
As Thin/Surface?	No
Target Bodies	All Bodies
Merge Topology?	Yes
Geometry Selection: 1	
Sketch	Sketch4

[4] Extrude-cut the material with **Through All** option. Click **Generate**. #

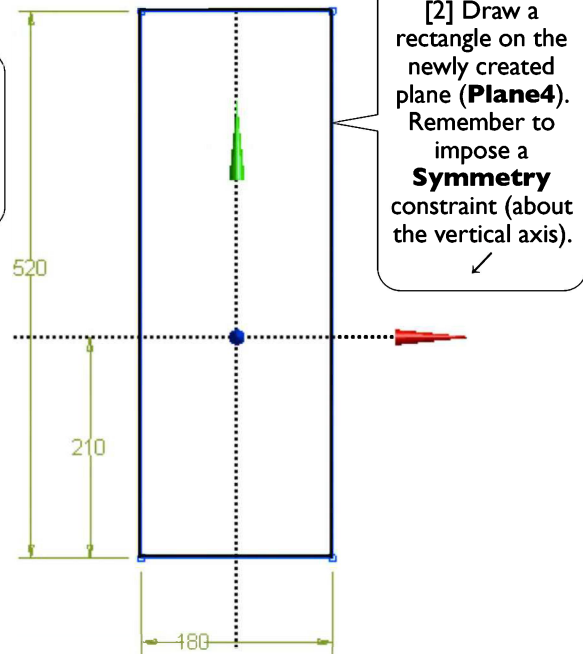


6.3.6 Create the Base

Details of Plane4

Plane	Plane4
Sketches	0
Type	From Plane
Base Plane	ZXPlane <input type="radio"/>
Transform 1 (RMB)	Offset Z <input type="radio"/>
FD1, Value 1	-170 mm <input type="radio"/>
Transform 2 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

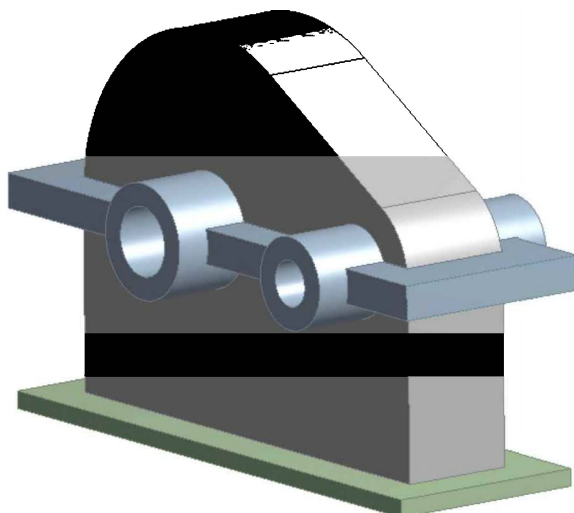
[1] Create a new plane (**Plane4**) by offsetting **ZXPlane** 170 mm downward. Click **Generate**. →



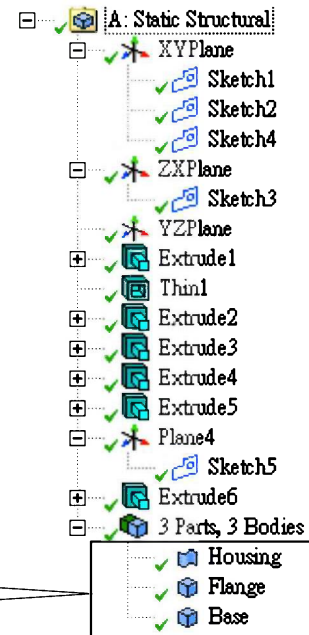
Details of Extrude6

Extrude	Extrude6
Geometry	Sketch5 <input type="radio"/>
Operation	Add Material
Direction Vector	None (Normal)
Direction	Reversed <input type="radio"/>
Extent Type	Fixed
FD1, Depth (>0)	15 mm <input type="radio"/>
As Thin/Surface?	No
Merge Topology?	Yes
Geometry Selection: 1	
Sketch	Sketch5

[3] Extrude 15 mm downward to create the base. Click **Generate**. ↓



[4] Rename each body for better readability. Close DesignModeler. #

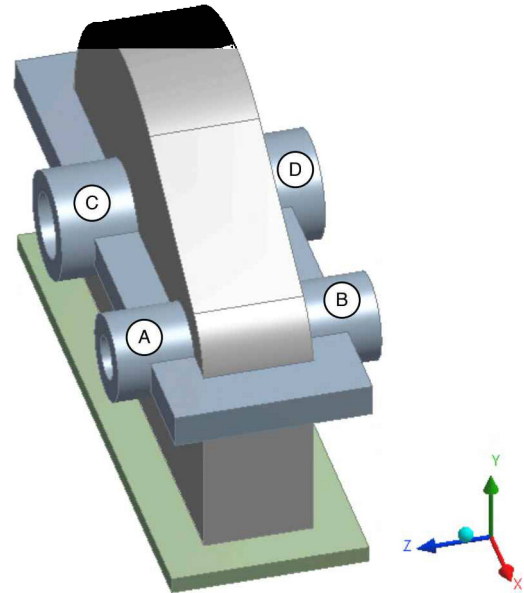


PART B. SIMULATION

6.3.7 Bearing Loads

[1] The bearing loads can be calculated according to the transmitted power, which is 175 hp in this case. Skipping the calculation details (if you are interested in the calculation details, please see the book by Zahavi^[Ref 1]), we summarize the bearing loads as follows. Note that these forces sum up to zero. ↓

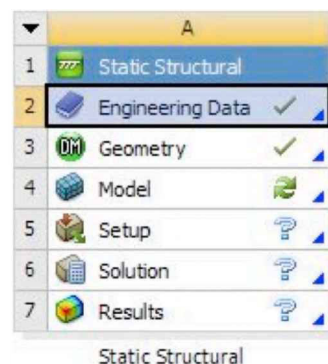
Bearing	F_x (N)	F_y (N)	F_z (N)
A	16000	30000	16000
B	6000	30000	0
C	3000	-30000	0
D	-25000	-30000	-16000



Specify Bearing Loads

[2] In **Mechanical**, a bearing load can be specified on a cylindrical surface. It is important to note that the bearing load is distributed on compressive side using projected area. This implies that axial components (here, the Z-components) will be zeros. In our case, the X- and Y-components (please refer to the global coordinates) of the bearing loads can be specified as bearing loads components, but the Z-components cannot be specified as bearing load components. The Z-components will be specified separately. #

6.3.8 Prepare Material Properties



[1] Click **Engineering Data**. We want to set up material properties. ↩

[2] Click **Engineering Data Sources**. →

[3] **Engineering Data Sources**. ↓

[4] Click **General Materials** to load a library of material properties. ↓

[5] Click the plus sign next to **Gray Cast Iron**. A "book" appears to the right, indicating that **Gray Cast Iron** has been added to **Engineering Data**. ↓

[6] Also click the plus sign next to **Stainless Steel**. ←

[7] Click **Engineering Data Sources** again to return to **Engineering Data**. ↓

1	A	B	C	D
	Data Source		Location	Description
2	★ Favorites			Quick access list and default items
3	General Materials			General use material samples for use in various analyses.
4	General Non-linear Materials			General use material samples for use in non-linear analyses.
5	Explicit Materials			Material samples for use in explicit analyses.
6	Hyperelastic Materials			Material samples for use in hyperelastic analyses.

1	A	B	C	D
	Contents of General Materials		Add	Source
2	Material			
3	Air			General
4	Aluminum Alloy			General
5	Concrete			General
6	Copper Alloy			General
7	Gray Cast Iron			General
8	Magnesium Alloy			General
9	Polyethylene			General
10	Silicon Anisotropic			General
11	Stainless Steel			General
12	Structural Steel			General
13	Titanium Alloy			General

[9] Click to return to **Project Schematic**. #

[8] Now, **Gray Cast Iron** and **Stainless Steel** are included in **Engineering Data**. ←

1	A	B	C	D
	Contents of Engineering Data		Source	Description
2	Material			
3	Gray Cast Iron			
4	Stainless Steel			
5	Structural Steel			Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material			

6.3.9 Assign Material and Thickness

Details of "Housing"

Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Thickness	3. mm
Thickness Mode	Manual
Offset Type	Middle
Behavior	None
Material	
Assignment	Stainless Steel
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	
Statistics	
CAD Attributes	
DMSheetThickness	

[3] Type 3 (mm) for **Thickness**. ↘

[2] Highlight **Housing** and select **Stainless Steel**. ↑

Details of "Flange"

Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Behavior	None
Material	
Assignment	Gray Cast Iron
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	
Statistics	

[4] Highlight **Flange** and select **Gray Cast Iron**. →

Details of "Base"

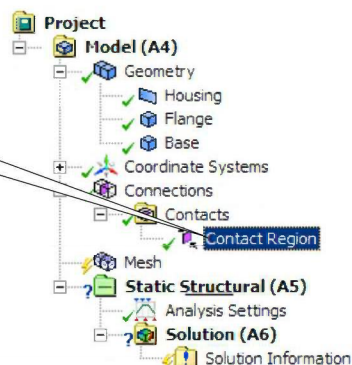
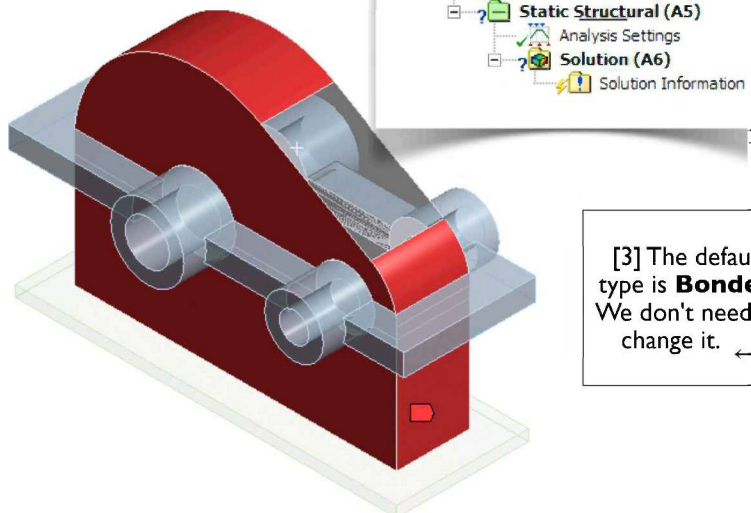
Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Behavior	None
Material	
Assignment	Gray Cast Iron
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	
Statistics	

[5] Highlight **Base** and select **Gray Cast Iron**. #

[1] Start up **Mechanical**. Make sure the unit system is **mm-kg-N-s**. Assign the material and thickness for each body [2-5]. ✓

6.3.10 Set Up Connections between Parts

[1] A contact region is detected and automatically created when the geometry was imported to **Mechanical**. Highlight **Contact Region**. →



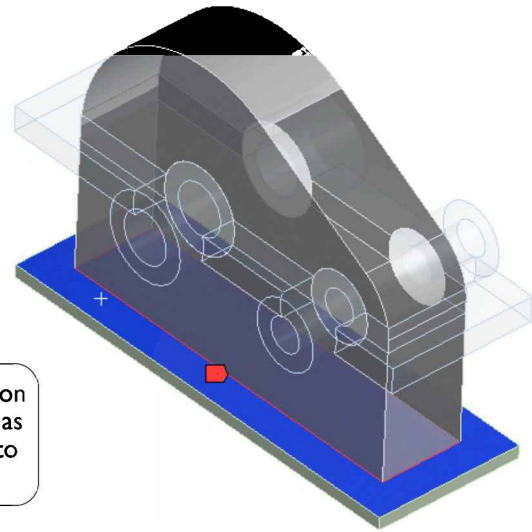
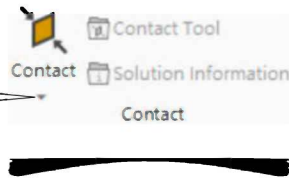
[2] The **Details** shows that the contact is established between the housing and the flange. The auto-detection works well. We don't need to change them. ↓

Detail of "Contact Region"

Scope	
Scoping Method	Geometry Selection
Contact	6 Faces
Target	4 Faces
Contact Bodies	Housing
Target Bodies	Flange
Contact Shell Face	Program Controlled
Shell Thickness Effect	No
Definition	
Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Turn Contact	Program Controlled
Turn Tolerance	1.7234 mm
Suppressed	No
Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Pinball Region	Program Controlled

[3] The default type is **Bonded**. We don't need to change it. ↵

[4] Contact between the housing and the base, however, is not detected. This is because **Mechanical** doesn't detect edge-to-surface contact automatically. We need to manually establish contact between the housing and the base. From the toolbars, select **Contact/Bonded**. ↓



Details of "Bonded - Housing To Base"	
Scope	
Scoping Method	Geometry Selection
Contact	4 Edges
Target	1 Face
Contact Bodies	Housing
Target Bodies	Base
Shell Thickness Effect	No
Definition	
Type	Bonded
Scope Mode	Manual
Tam Contact	Program Controlled
Suppressed	No
Advanced	
Formulation	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Pinball Region	Program Controlled

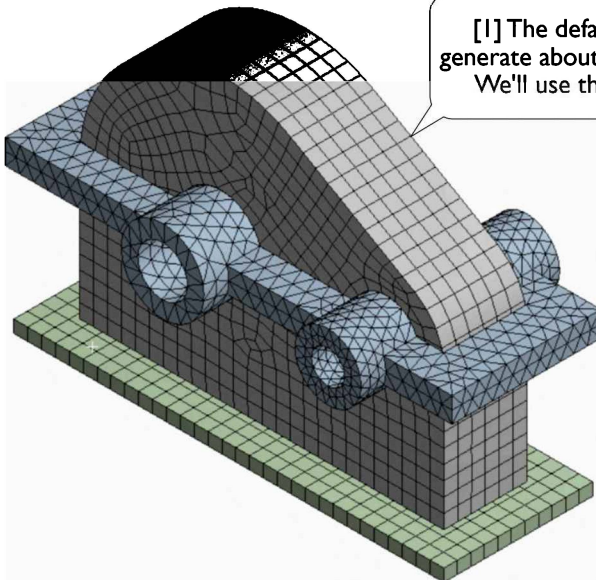
[5] Control-select 4 edges on the bottom of the housing as **Contact**. You may need to use the edge filter. ↓

[6] Select top face of the base as **Target**. Use the face filter. ↓

Do contacts always introduce nonlinearity?

[7] It depends on the type of contacts. In this case, the **Bonded** contacts do not introduce any nonlinearities. For details, see 13.1.8, page 476. #

6.3.11 Generate Mesh



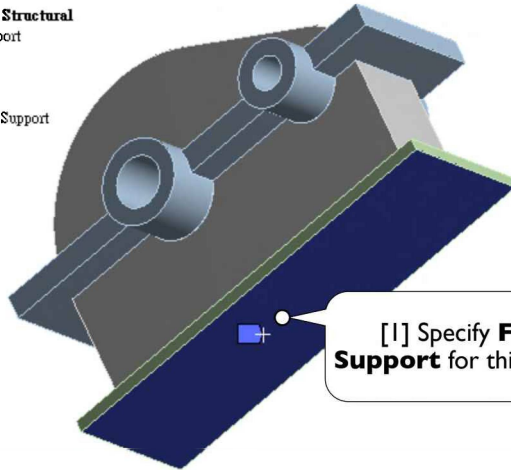
[1] The default settings generate about 16000 nodes. We'll use this mesh. #

Details of "Mesh"	
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
<input type="checkbox"/> Element Size	Default (16.83 mm)
Sizing	
Use Adaptive Sizing	No
Use Uniform Size Function For Sheets	No
<input type="checkbox"/> Growth Rate	Default (1.2)
<input type="checkbox"/> Max Size	Default (16.83 mm)
Mesh Defeaturing	Yes
<input type="checkbox"/> Defeature Size	Default (8.4149e-002 mm)
Capture Curvature	Yes
<input type="checkbox"/> Curvature Min Size	Default (0.1683 mm)
<input type="checkbox"/> Curvature Normal Angle	Default (30.0°)
Capture Proximity	No
Bounding Box Diagonal	689.37 mm
Average Surface Area	16127 mm²
Minimum Edge Length	15.0 mm
Quality	
Inflation	
Batch Connections	
Advanced	
Statistics	
<input type="checkbox"/> Nodes	16304
<input type="checkbox"/> Elements	8586

6.3.12 Specify Support

A: Static Structural
Fixed Support
Time: 1. s

Fixed Support



[1] Specify **Fixed Support** for this face. →

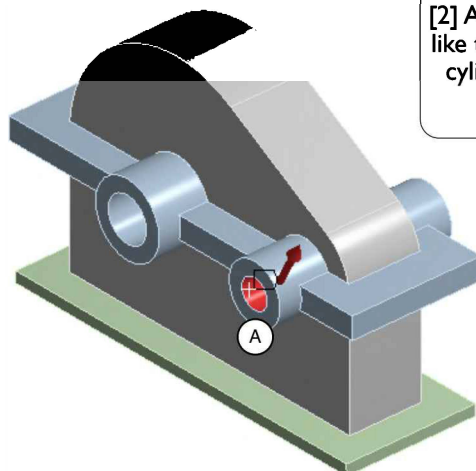
Support Conditions

[2] In this case, a **Compression Only Support** or a **Frictionless Support** would be more realistic than a **Fixed Support**. However, the results will be the same for these three support conditions. Since the external forces sum up to zero, the support reaction is zero. **Fixed Support** in this case has an advantage: it doesn't introduce nonlinearities or the need for weak springs (3.1.8[7], page 116). **Compression Only Support** would introduce nonlinearities, and **Frictionless Support** would introduce the need for weak springs. #

6.3.13 Apply Bearing Loads

Details of "Bearing Load A"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Bearing Load
Define By	Components
Coordinate System	Global Coordinate System
X Component	16000 N
Y Component	30000 N
Z Component	0. N
Suppressed	No

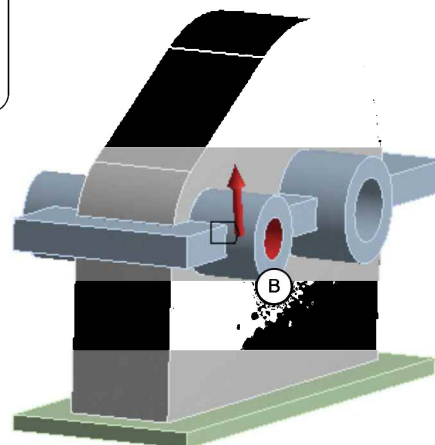
Bearing Load A: 34000 N
Components: 16000, 30000, 0. N



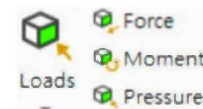
[2] Apply a bearing load like this on the internal cylindrical surface of bearing A. →

Details of "Bearing Load B"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Bearing Load
Define By	Components
Coordinate System	Global Coordinate System
X Component	6000. N
Y Component	30000 N
Z Component	0. N
Suppressed	No

Bearing Load B: 30594 N
Components: 6000, 30000, 0. N



[4] Apply a bearing load like this on the internal cylindrical surface of bearing B. ↘




[1, 3, 5, 7] Select **Loads/Bearing Load**. ← ↗ ↘

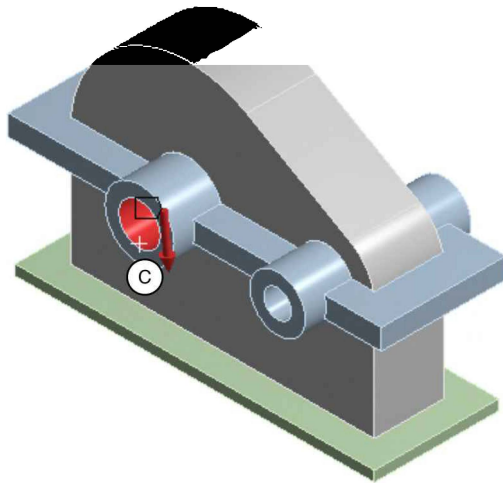
Details of "Bearing Load C"		
Scope		
Scoping Method	Geometry Selection	
Geometry	1 Face	<input type="radio"/>
Definition		
Type	Bearing Load	
Define By	Components	<input type="radio"/>
Coordinate System	Global Coordinate System	
X Component	3000. N	<input type="radio"/>
Y Component	-30000 N	<input type="radio"/>
Z Component	0. N	<input type="radio"/>
Suppressed	No	


[6] Apply a bearing load like this on the internal cylindrical surface of bearing C. ([7] is on the last page.)

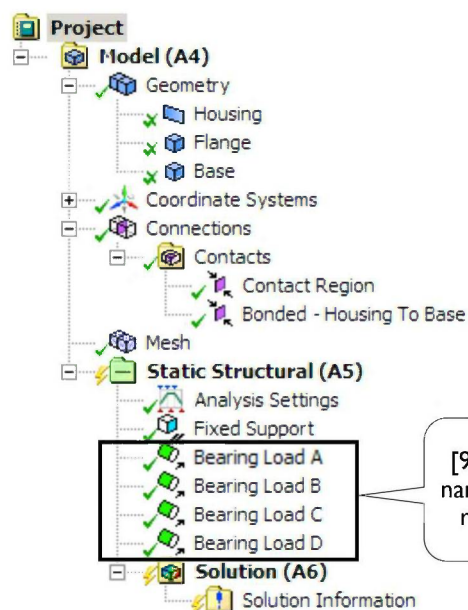
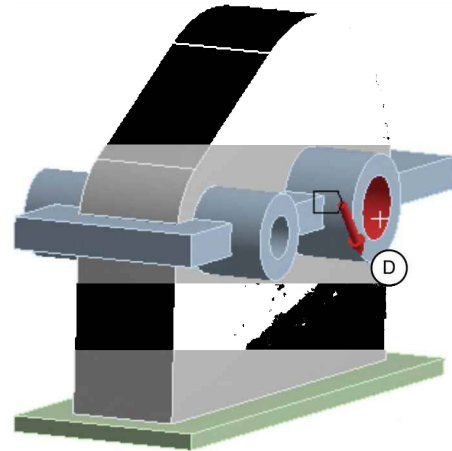
Details of "Bearing Load D"		
Scope		
Scoping Method	Geometry Selection	
Geometry	1 Face	<input type="radio"/>
Definition		
Type	Bearing Load	
Define By	Components	<input type="radio"/>
Coordinate System	Global Coordinate System	
X Component	-25000 N	<input type="radio"/>
Y Component	-30000 N	<input type="radio"/>
Z Component	0. N	<input type="radio"/>
Suppressed	No	

[8] Apply a bearing load like this on the internal cylindrical surface of bearing D. ↓

 Bearing Load C: 30150 N
Components: 3000, -30000, 0. N



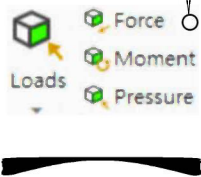
 Bearing Load D: 39051 N
Components: -25000, -30000, 0. N



[9] Change the names for better readability. #

6.3.14 Apply Axial Loads

[1, 3] Select **Force**.



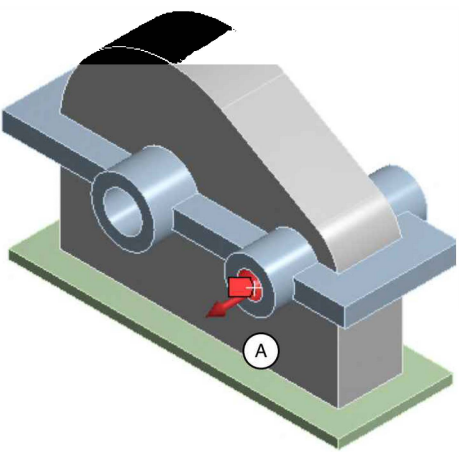
[2] Apply a force on the internal cylindrical surface of bearing A. ←

Details of "Axial Load at A"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Force
Define By	Components <input type="radio"/>
Coordinate System	Global Coordinate System
X Component	0. N (samped)
Y Component	0. N (samped)
Z Component	16000 N (samped) <input type="radio"/>
Suppressed	No

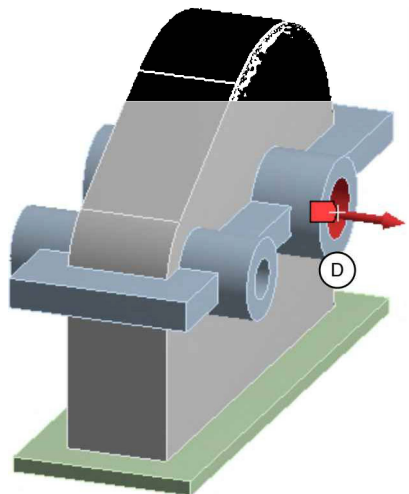
[4] Apply a force on the internal cylindrical surface of bearing D. ✓

Details of "Axial Load at D"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Force
Define By	Components <input type="radio"/>
Coordinate System	Global Coordinate System
X Component	0. N (samped)
Y Component	0. N (samped)
Z Component	-16000 N (samped) <input type="radio"/>
Suppressed	No

■ Axial Load at A: 16000 N
Components: 0, 0, 16000 N



■ Axial Load at D: 16000 N
Components: 0, 0, -16000 N




Project

- Model (A4)
 - Geometry
 - Housing
 - Flange
 - Base
 - Coordinate Systems
 - Connections
 - Contacts
 - Contact Region
 - Bonded - Housing To Base
 - Mesh
 - Static Structural (A5)
 - Analysis Settings
 - Fixed Support
 - Bearing Load A
 - Bearing Load B
 - Bearing Load C
 - Bearing Load D
 - Axial Load at A
 - Axial Load at D
 - Solution (A6)
 - Solution Information
 - Total Deformation
 - Equivalent Stress

[5] Change names for better readability. ↓

[6] Insert two results objects. →

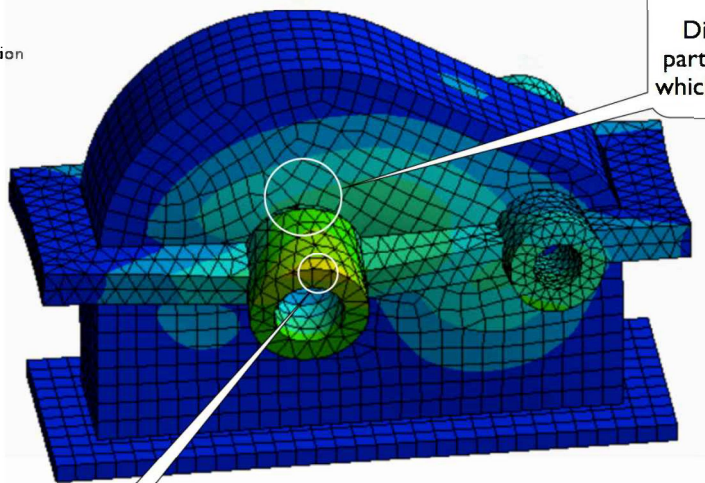
[7] Solve the model. #



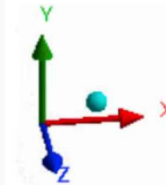
6.3.15 View the Results

A: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1

0.73794 Max
0.65594
0.57395
0.49196
0.40997
0.32797
0.24598
0.16399
0.081993
0 Min



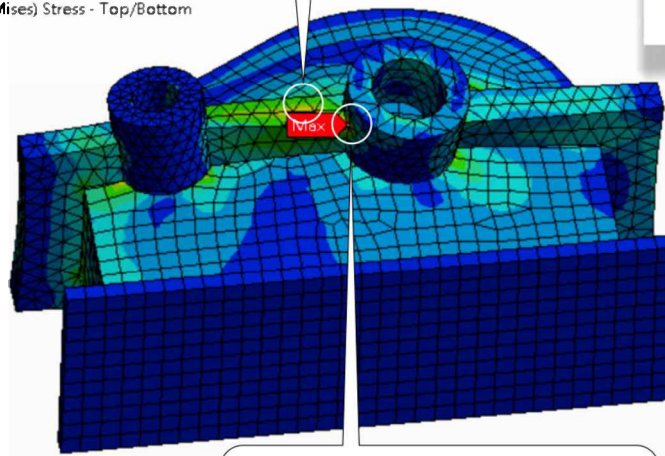
[1] The gap here is not realistic. This is due to the exaggeration of the deformation. Remember that each part is treated independently. Displacements at boundary of two parts may have small numerical errors, which become a gap when enlarged. ✓



[2] The maximum deformation is 0.74 mm. ✓

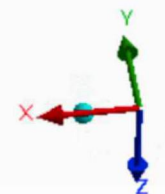
A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress - Top/Bottom
Unit: MPa
Time: 1

181.86 Max
161.66
141.45
121.25
101.04
80.832
60.626
40.419
20.213
0.0060908 Min



[4] The engineer should use the stresses that are not singular for design purposes. ✓

Probe
Maximum
Minimum



Wrap Up

[5] Save the project and exit Workbench. #

[3] The maximum stress is 182 MPa, occurring at a sharp corner. It is not realistic either. It is a singular stress (PART E, Section 3.5, pages 166-169). ↑

Reference

1. Zahavi, E., *The Finite Element Method in Machine Design*, Prentice-Hall, 1992; Chapter 10. Gear Box.

Section 6.4

Review

6.4.1 Keywords

Choose a letter for each keyword, from the list of descriptions

1. () Auto-Detection of Contact
2. () Shell Elements
3. () Top/Bottom of Surface Body

Answers:

1. (C) 2. (A) 3. (B)

List of Descriptions

(A) A planar (2D) element that can be arranged in the 3D space. It is used to mesh a body when one of its dimensions is much smaller than the other two dimensions. Each node has 6 degrees of freedom: 3 translational and 3 rotational. Due to the presence of rotational degrees of freedom, it is very efficient to model the problems dominated by the out-of-plane bending modes, contrasting to a solid element, which does not have rotational degrees of freedom.

(B) Each surface body has a top side and a bottom side. When you select a surface body, only the top side is highlighted. Loads are applied on the top side. By default, results are reported on both sides.

(C) When a geometry attaches to **Mechanical**, it automatically detects and establishes possible contacts between parts, wherever the gaps between parts are less than a tolerance. The contact type is **Bonded** by default.

6.4.2 Questions

Surface Body versus Thin Solid Body

Q: In 6.3.3[5] (page 257), we type a zero value for the thickness to create a surface body. If we provide a nonzero value, it would create a "thin" solid body instead of a surface body. What is an essential difference between a surface body and a thin solid body?

A: A surface body will be meshed with shell elements (1.3.3[10-11], page 38), while a thin solid body will be meshed with solid elements (1.3.3[2-5], page 38).

Shell Elements versus Solid Elements

Q: What is the essential difference between shell elements and solid elements?

A: The shell elements have rotational degrees of freedom (1.3.3[10-11], page 38), while solid elements do not (1.3.3[2-5], page 38).

Triangular Plate

Q: Can we perform a simulation with a 3D surface model for the triangular plate in Section 3.1? If positive, are there any advantages of doing that over the 2D solid model?

A: Yes, we could, but there is no advantage over the 2D solid model. Remember that 3D surface bodies are meshed with shell elements. The essential difference between shell elements and 2D solid elements is that shell elements can have out-of-plane deformation (warping) while 2D solid elements cannot. In the case of triangular plate, there is no out-of-plane deformation. There is no need to use shell elements. The above discussions also apply on the spur gear of Section 3.4 and the filleted bar of Section 3.5.

Axisymmetric Bodies

In the beginning and the end of Section 6.1, we mentioned that we could model the bellows using axisymmetric 2D solid body or 2D line body. Make sure that you do understand the meaning of the statement.

Chapter 7

Line Models

Many real-world objects can be modeled as line bodies. When a body has small lateral dimensions and has a uniform cross-section, it is often modeled as a line body. The most obvious applications of line models are frame, beam, and truss structures. Workbench meshes a line body with beam elements (1.3.3[13], page 39). Advantages of using line models over surface models or solid models include: (a) creating line models is usually easier, (b) the problem size is much smaller, and (c) the solution can be more accurate. Therefore, engineers should consider using line models instead of surface or solid models whenever possible. Workbench stores many built-in cross sections in the database to be chosen from by the users. Workbench allows the rendering of cross-sections when displaying the model, so that the line bodies visually look like solid bodies.

Purpose of This Chapter

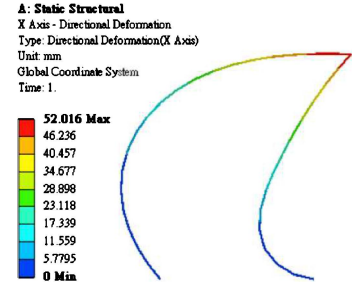
The main purpose of this chapter is to introduce the use of beam elements (1.3.3[13], page 39). This chapter guides the students to create line models and perform simulations using these models. The chapter provides three examples; two of them are entirely line models, while the other one is a model mixed up with line bodies and surface bodies.

About Each Section

Section 7.1 creates a flexible gripper model and performs a simulation using the model. Section 7.2 demonstrates the creation of a truss structure and the simulation. Section 7.3 uses a two-story building as an example to demonstrate the creation of a building structure and the simulation. Another purpose of Section 7.3 is to demonstrate how surface bodies and line bodies can be mixed up in a simulation model.

Section 7.1

Flexible Gripper^[Ref 1]



7.1.1 About the Flexible Gripper

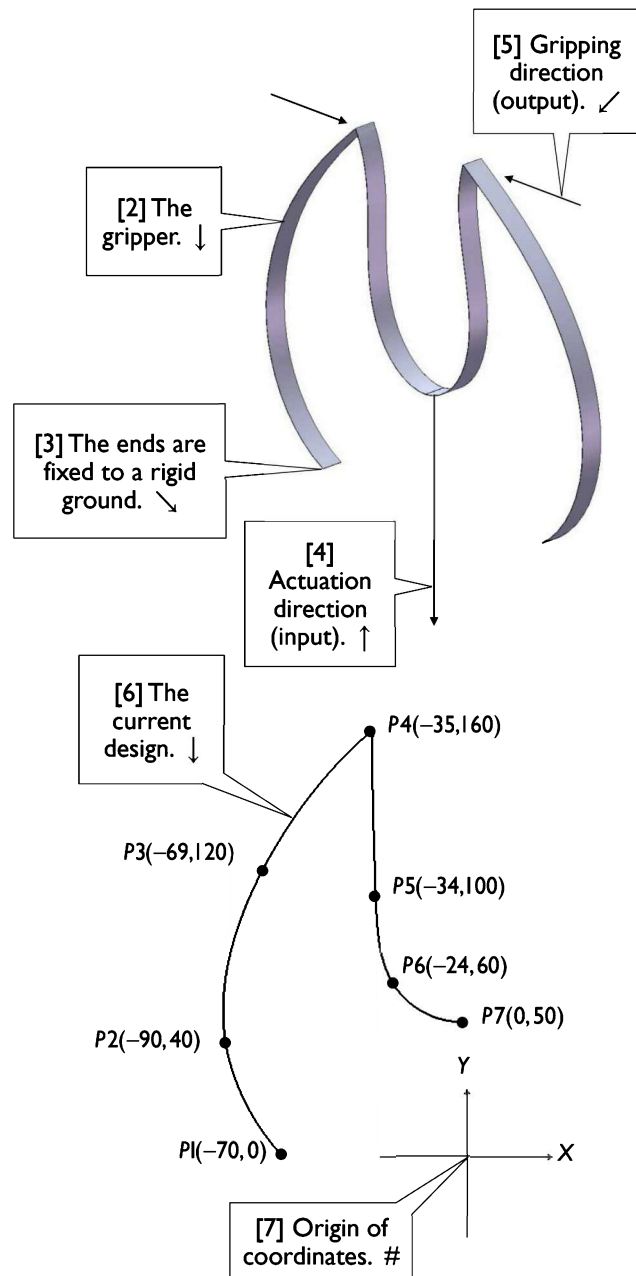
[1] The gripper [2-5] is made of a POM (polyoxymethylene, a plastic polymer), which has a Young's modulus of 2 GPa and a Poisson's ratio of 0.35. It has a rectangular cross section of 1x5 mm².

A concern in designing this gripper is its geometric advantage (GA). The GA is defined as the ratio of the horizontal output displacement [5] to the input actuation [4]. The GA value is used to assess the efficiency of the gripper; the larger the better. The main purpose of this simulation is to assess the GA value of the current design as shown in the figure [6]. Note that only half of the gripper is modeled due to the symmetry.

The profile consists of two smooth spline curves defined by 7 key points, whose numbering and coordinates are shown in the figure [6-7].

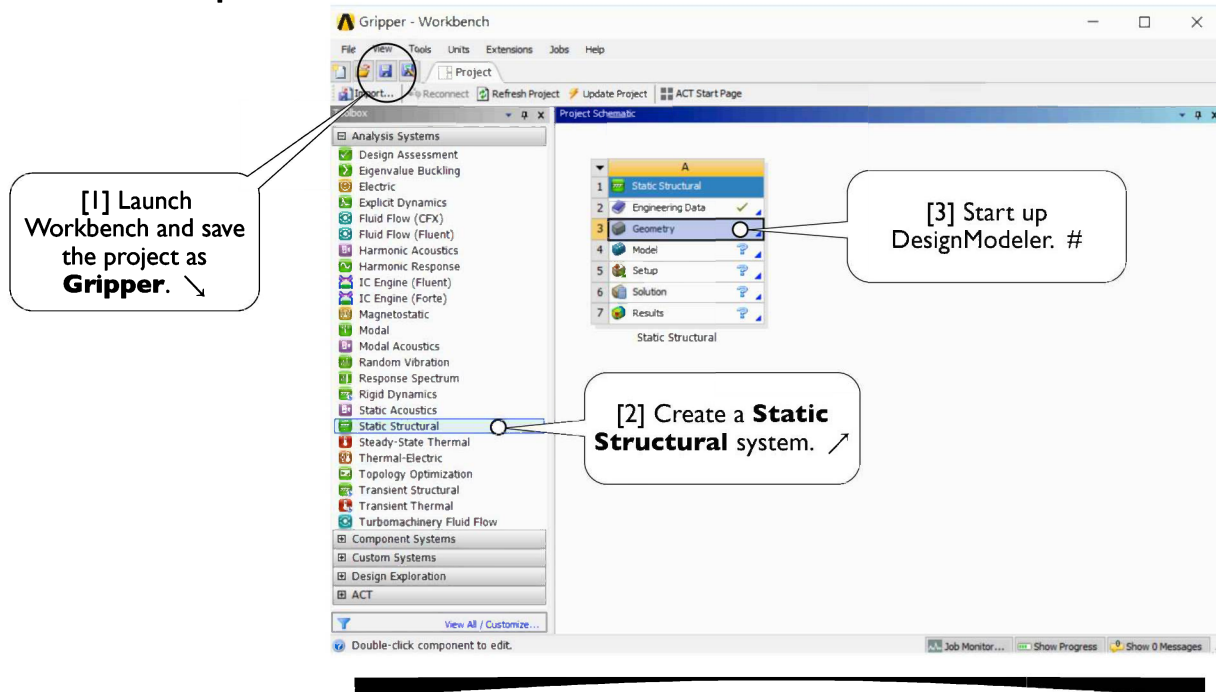
The problem could be solved with a 2D simulation. With 2D simulations, however, the current version of **Mechanical** supports only solid models (it doesn't support 2D surface model or line model). Compared with a 3D line model, a 2D solid model is not efficient (in terms of accuracy and computing time.) We decide to go for a 3D line model although the geometry and the motion are entirely on a plane.

We will create a line model in the first part of this section. The model will be used in the second part of this section to simulate the motion of gripping and to assess the GA value. The model will be used again in Section 8.1 to demonstrate an optimization capability of Workbench, in which we want to relocate the positions of the key points to achieve an optimum GA value. ↗

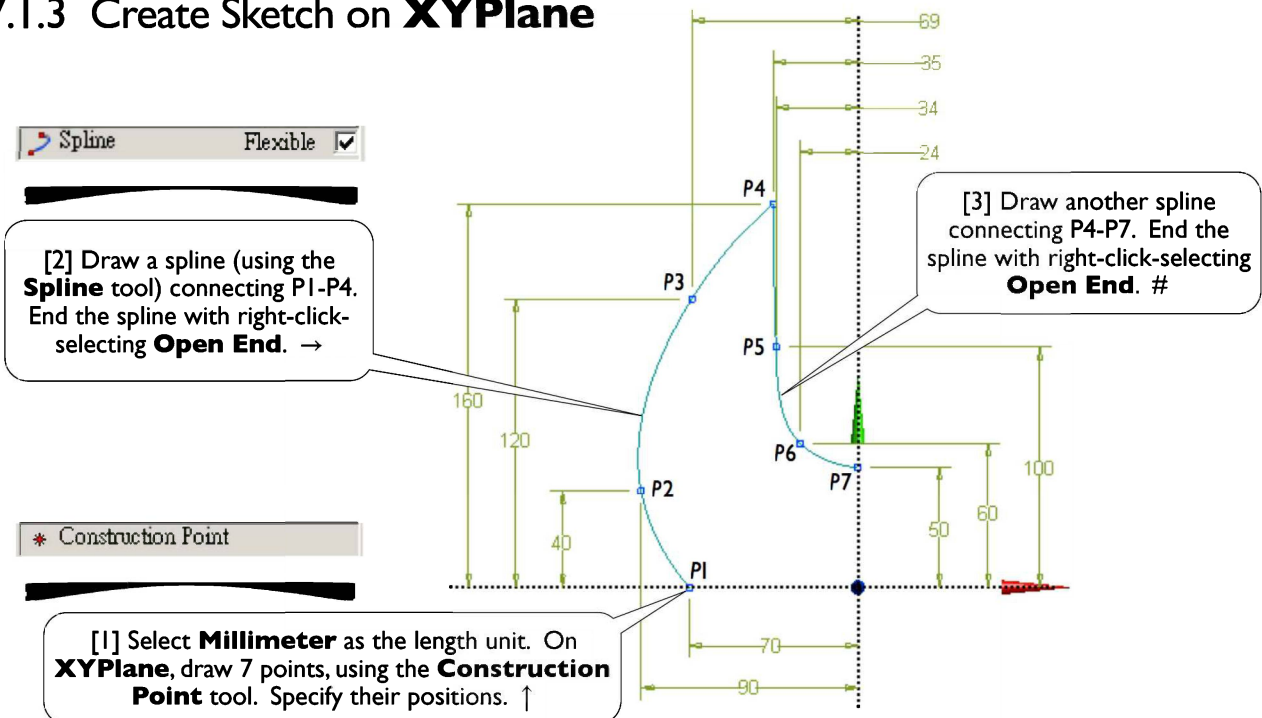


PART A. GEOMETRIC MODELING

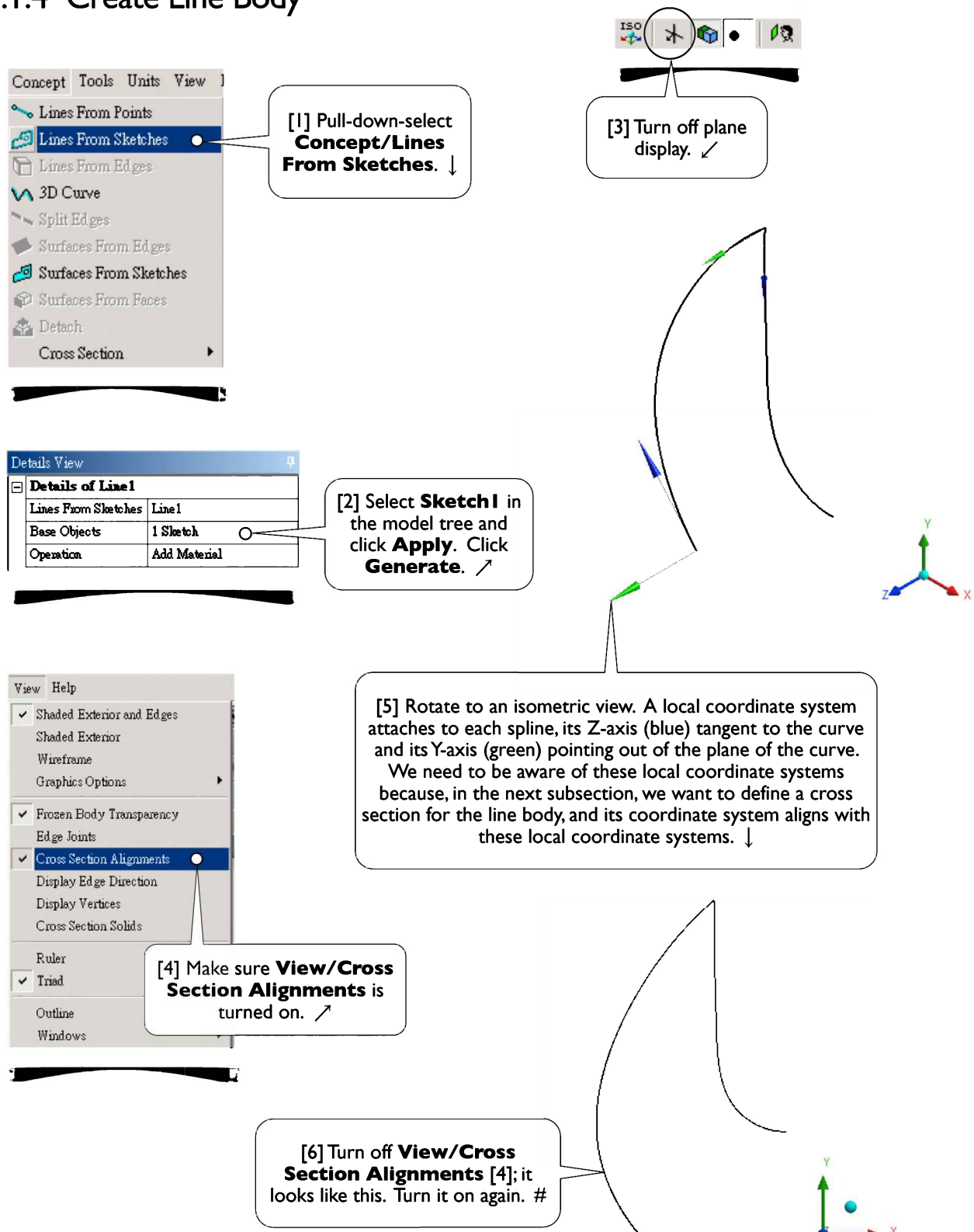
7.1.2 Start Up



7.1.3 Create Sketch on **XYPlane**



7.1.4 Create Line Body



[1] Pull-down-select **Concept/Lines From Sketches**. ↓

[2] Select **Sketch1** in the model tree and click **Apply**. Click **Generate**. ↗

[3] Turn off plane display. ↙

[4] Make sure **View/Cross Section Alignments** is turned on. ↗

[5] Rotate to an isometric view. A local coordinate system attaches to each spline, its Z-axis (blue) tangent to the curve and its Y-axis (green) pointing out of the plane of the curve. We need to be aware of these local coordinate systems because, in the next subsection, we want to define a cross section for the line body, and its coordinate system aligns with these local coordinate systems. ↓

[6] Turn off **View/Cross Section Alignments** [4]; it looks like this. Turn it on again. #

7.1.5 Create a Rectangular Cross Section

Details View

Details of Rect1		
Sketch	Rect1	
Show Constraints?	No	
Dimensions: 2		
B	1 mm	
H	5 mm	
Edges: 4		
Line	La11	
Line	La12	
Line	La13	
Line	La14	
Physical Properties: 10		
A	5 mm²	
I _{xx}	10.417 mm⁴	
I _{xy}	0 mm⁴	
I _{yy}	0.41667 mm⁴	
I _w	0.73342 mm⁶	
J	1.4571 mm⁴	

[2] Type 1 (mm) for **B** and 5 (mm) for **H**. ↓

Concept Tools Units View

- Lines From Points
- Lines From Sketches
- Lines From Edges
- 3D Curve
- Split Edges
- Surfaces From Edges
- Surfaces From Sketches
- Surfaces From Faces
- Detach

Cross Section

- Rectangular**
- Circular
- Circular Tube
- Channel Section
- I Section
- Z Section
- L Section
- T Section
- Hat Section
- Rectangular Tube
- User Integrated
- User Defined

[1] Pull-down-select **Concept/Cross Section/Rectangular**. ←

[3] A rectangular cross section **Rect1** is created. Note that its local Y-axis (green) is in the longer direction; that is exactly what we intend (see 7.1.4[5], last page). You don't need to click **Generate** when creating a cross section. #

7.1.6 Assign the Cross Section to the Line Body

A: Static Structural

- XYPlane
- Sketch1
- ZXPlane
- YZPlane
- Line1
- 1 Cross Section
- Rect1
- 1 Part, 1 Body
- Line Body

[1] Select **Line Body**. ↓

Details View

Details of Line Body	
Body	Line Body
Faces	0
Edges	2
Vertices	3
Cross Section	Rect1
Offset Type	Centroid
Shaded Topology Method	Edge Joints
Geometry Type	DesignModeler

[2] Select the newly created cross section. ↓

View Help

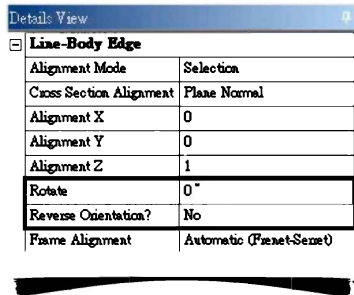
- Shaded Exterior and Edges
- Shaded Exterior
- Wireframe
- Graphics Options
- Frozen Body Transparency
- Edge Joints
- Cross Section Alignments
- Display Edge Direction
- Display Vertices
- Cross Section Solids**
- Blade Ruling Lines
- Ruler
- Triad
- Outline

[3] To display the cross section, turn on **View/Cross Section Solids**. ↑

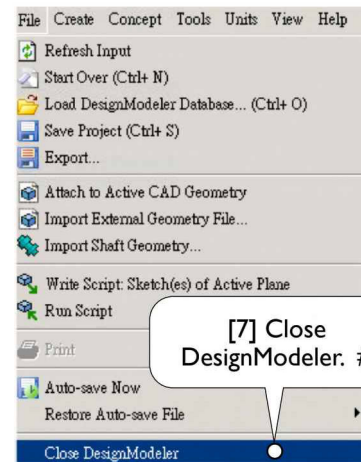
[4] The rectangular shape of the cross section is now visualized. Note that this is for visual effect only. This is a line body, not a solid body. ↩

Adjusting Cross Section Alignments

[5] When creating line models, make sure the cross section alignments are correct. In our case, the default alignments are what we want, so we don't need to make any adjustment. In other cases, if adjustment of cross section alignments is needed, you can turn on the edge selection filter, select the edges of the line bodies, and type the rotation angle [6]. We will demonstrate the procedure in 7.3.9, page 300. ↓



[6] Cross section alignment can be adjusted by rotating an angle. →



[7] Close DesignModeler. #

PART B. SIMULATION

7.1.7 Prepare Material Properties

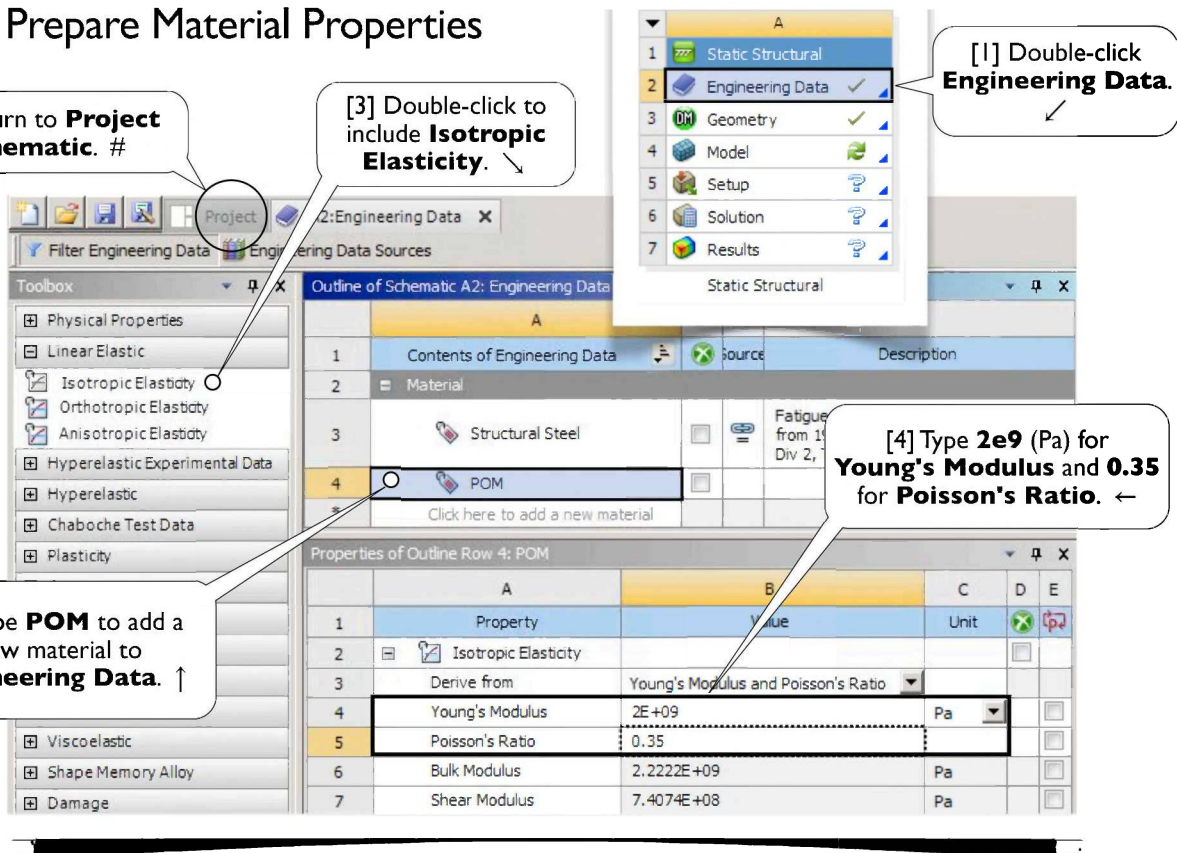
[5] Return to **Project Schematic**. #

[3] Double-click to include **Isotropic Elasticity**. ↓

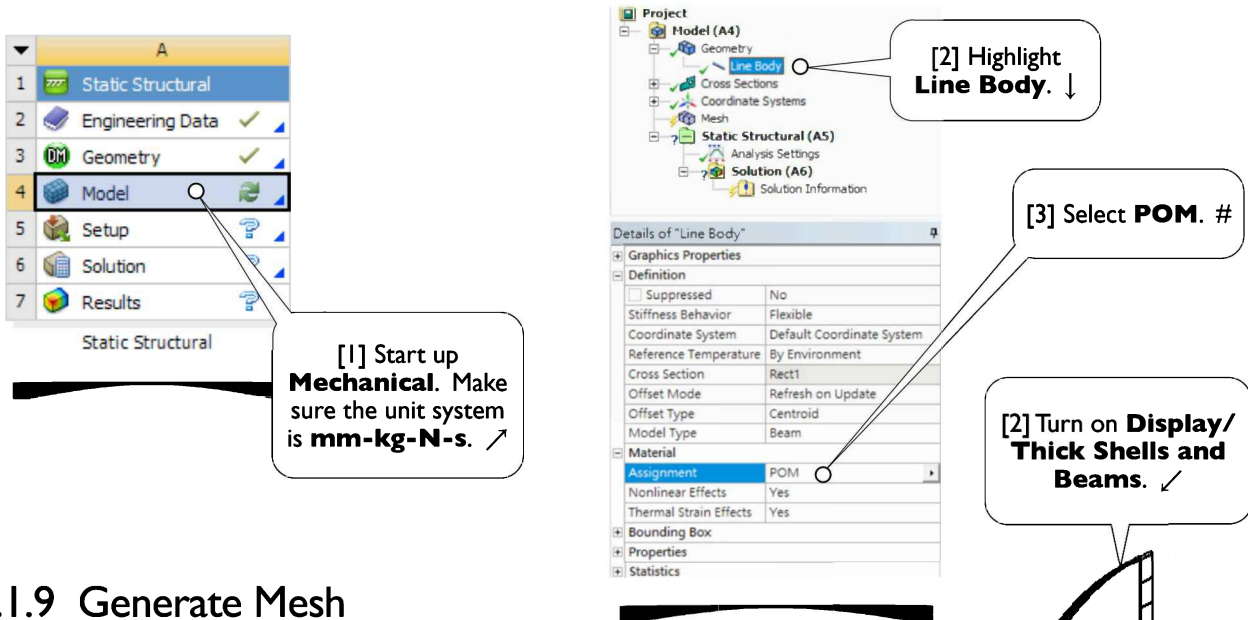
[1] Double-click **Engineering Data**. ↙

[2] Type **POM** to add a new material to **Engineering Data**. ↑

[4] Type **2e9 (Pa)** for **Young's Modulus** and **0.35** for **Poisson's Ratio**. ←



7.1.8 Start Up **Mechanical** and Assign Material



[1] Start up **Mechanical**. Make sure the unit system is **mm-kg-N-s**. ↗

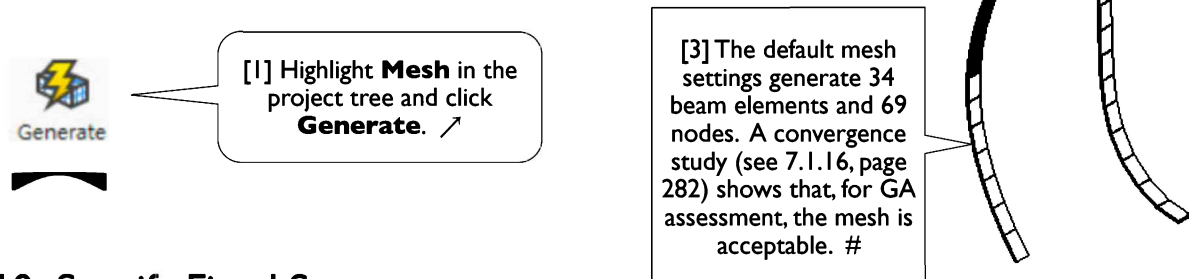
[2] Highlight **Line Body**. ↓

[3] Select **POM**. #

[2] Turn on **Display/Thick Shells and Beams**. ✓

Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Cross Section	Rect1
Offset Mode	Refresh on Update
Offset Type	Centroid
Model Type	Beam
Material	
Assignment	POM
Nonlinear Effects	Yes
Thermal Strain Effects	Yes

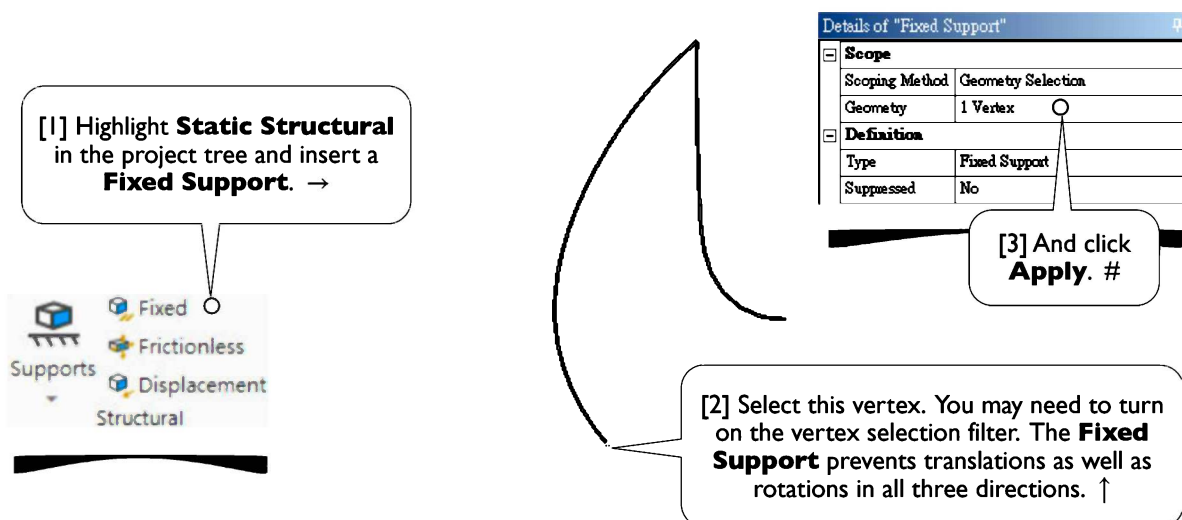
7.1.9 Generate Mesh



[1] Highlight **Mesh** in the project tree and click **Generate**. ↗

[3] The default mesh settings generate 34 beam elements and 69 nodes. A convergence study (see 7.1.16, page 282) shows that, for GA assessment, the mesh is acceptable. #

7.1.10 Specify Fixed Supports



[1] Highlight **Static Structural** in the project tree and insert a **Fixed Support**. →

[2] Select this vertex. You may need to turn on the vertex selection filter. The **Fixed Support** prevents translations as well as rotations in all three directions. ↑

[3] And click **Apply**. #

Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Fixed Support
Suppressed	No

7.1.1.1 Specify Symmetry Condition and the Actuation

[1] Select **Supports/Fixed Rotation**. →

[2, 6] Select this vertex. ↗ ↓

[3] And click **Apply**. ↓

Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Fixed Rotation
Coordinate System	Global Coordinate System
Rotation X	Free
Rotation Y	Fixed
Rotation Z	Fixed
Suppressed	No

[4] Set **Rotation X** free and leave the rotations in Y- and Z-direction fixed. ✓

[5] Click **Displacement**. ↗

Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	0. mm (ramped)
Y Component	-50. mm (ramped)
Z Component	Free
Suppressed	No

[7] And click **Apply**. →

[8] Type 0 to fix the displacement in X-direction. This condition together with those in [4] ensure the symmetry about **YZPlane** (see [10]). ↓

[9] Apply 50 mm displacement downward. ↓

Symmetry Conditions for Shell and Beam Elements

[10] The purpose of the steps [1-8] is to set up a symmetry condition about **YZPlane**, while step [9] is to set up a downward actuation.

Beam elements, like shell elements, have rotational degrees of freedom (1.3.3[13], page 39). The rule of symmetry conditions for shell and beam elements is this: fixing (zero values) the **out-of-plane translations** and the **in-plane rotations**.

Consider the boundary conditions at the vertex shown in [2]. In this case, **YZPlane** is the plane of symmetry, X-displacement is the out-of-plane translation [8], Y-rotation and Z-rotation are the in-plane rotations [4].

With surface models, you may set up symmetry conditions using the **Symmetry** tool (3.1.1[7], page 119). Even so, make sure you know Workbench sets up symmetry conditions by fixing the out-of-plane translations and the in-plane rotations. As a good engineer, use software only when you know how it works.

The **Symmetry** tool (3.1.1[7], page 119) is not applicable for a line model. #

7.1.12 Set Up **Analysis Settings**

[1] Highlight **Analysis Settings**. ↓

Details of "Analysis Settings"	
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Off
Define By	Substeps
Number Of Substeps	10.
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Restart Controls	
Nonlinear Controls	
Output Controls	
Analysis Data Management	
Visibility	

[2] Turn off **Auto Time Stepping** and type 10 for **Number Of Substeps**. ↓

[3] Turn on **Large Deflection**. →

Why Large Deflection?

[4] Turning on large deflection is to include geometry nonlinearity, which always gives more accurate solutions but takes more computing time. To justify the inclusion of geometry nonlinearity, at the end of this section, turn off **Large Deflection** and rerun this case. A substantial difference in the results is an indication that the inclusion of geometry nonlinearity is necessary.

Auto Time Stepping

This model is simple enough that we actually don't need to change any default settings other than just turning on **Large Deflection**. The solution would be complete in just 3 substeps, with auto time stepping controlled by the program.

The reason we turn off **Auto Time Stepping** and specify 10 substeps is because we want to gather more data for plotting an input-displacement-versus-output-displacement chart (see 7.1.15[7], page 281). #

7.1.13 Set Up **Solution** Branch and Solve the Model

[1] Highlight **Solution**. →

[2] Select **Deformation/ Total**. →

[3] Select **Toolbox/Beam Tool**. ✓

[4] The **Solution** branch looks like this. →

[5] Click **Solve**. #

7.1.14 View Results

[1] Ignore the warning message. ✓

[2] Highlight **Total Deformation**. ✓

[3] The maximum total displacement is 77.153 mm. ↓

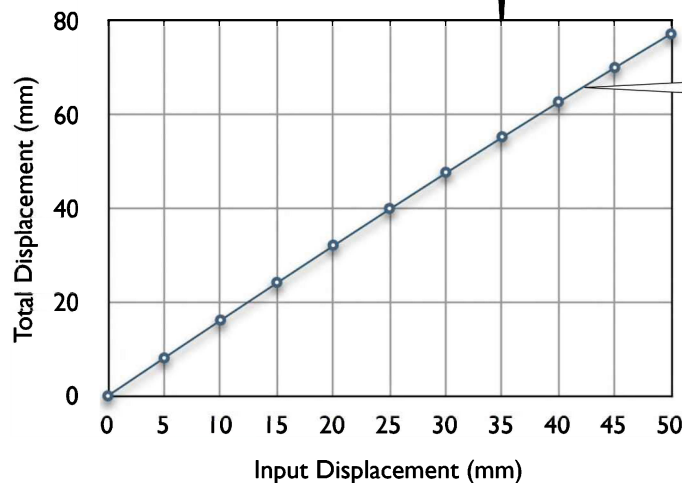
[4] Click **Result Sets**. ✓

[5] Set **Result** to **True Scale** and click **Play**. Click again to pause the animation. →

[6] Numerical data for each substep are available. You could copy/paste to your spreadsheet for further processing. ✓

[7] A graph of maximum-response-versus-time is displayed here. We reproduce the graph as shown in [8-9]. ↘

Time [s]	Minimum [mm]	Maximum [mm]
1 0.1	0.	8.0845
2 0.2	0.	16.164
3 0.3	0.	24.189
4 0.4	0.	32.126
5 0.5	0.	39.954
6 0.6	0.	47.66
7 0.7	0.	55.236
8 0.8	0.	62.68
9 0.9	0.	69.986
10 1.	0.	77.153



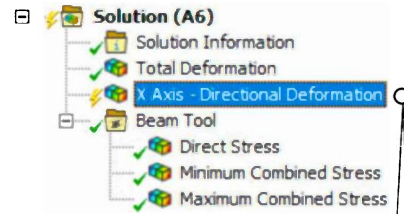
[9] The total-displacement-versus-input-displacement curve is almost a straight line. #

[8] The maximum input displacement (50 mm) is divided into 10 substeps; therefore, each substep is 5 mm of input displacement. ↑

7.1.15 Assess GA Value



[1] With **Solution** in the project tree highlighted, select **Deformation/Directional**. ↓



[3] Right-click **Directional Deformation** and rename it like this. ↓



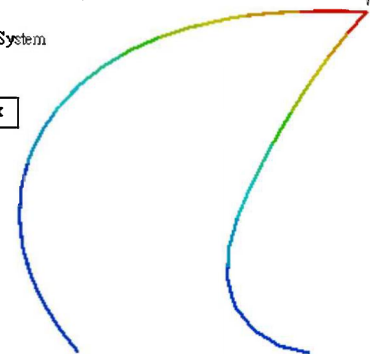
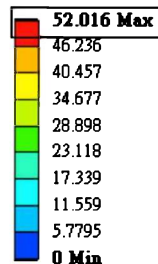
[4] Solve. ↓

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Directional Deformation
Orientation	X Axis
By	Time
Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
Results	
Minimum	
Maximum	
Minimum Occurs On	
Maximum Occurs On	
Information	

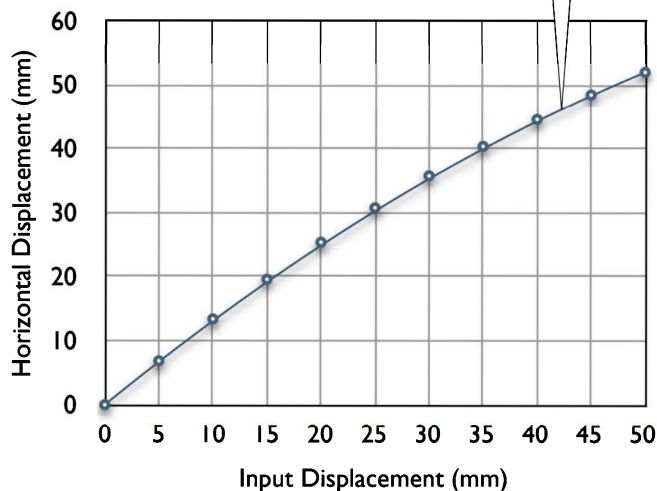
[2] You don't need to make any change in the details. We want to see displacements in **X Axis**. →

[5] The maximum horizontal displacement is 52.016 mm. ↓

A: Static Structural
X Axis - Directional Deformation
Type: Directional Deformation(X Axis)
Unit: mm
Global Coordinate System
Time: 1.



[7] The **Graph** window shows a horizontal-displacement-versus-input-displacement plot. #



Horizontal-Displacement-versus-Input-Displacement Plot

[6] A horizontal-displacement-versus-input-displacement curve is shown as [7]. The curve is nearly but not a straight line. The GA value in this case is about 1.04 (52/50). ↖

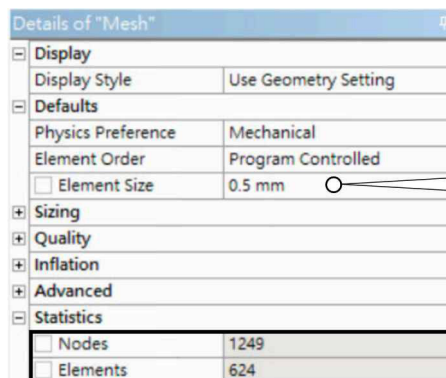
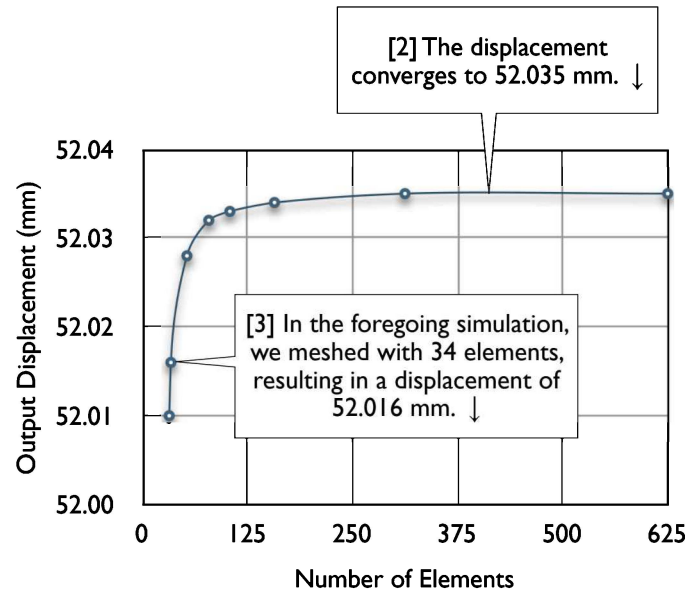
7.1.16 Convergence Study of Beam Elements

[1] A convergence study of the gripper model is summarized in the table/figure below [2-3]. The table/figure shows that the displacement converges to 52.035 mm [2]. In the foregoing simulation, we meshed with 34 elements using the default settings and obtained a displacement of 52.016 mm [3]. That is accurate enough for GA assessment, but may not be adequate for other purposes (e.g., stress assessment).

Generally, models meshed with beam elements or shell elements converge very fast. The meshing consideration is mostly geometric. In our case, since it is a curved structure, the major consideration is to mesh the structure so that the geometry of the meshed finite element model doesn't deviate from the original geometry too much.

For a structure composed by straight beams, we can obtain a solution equal to theoretical values by meshing each straight beam with a single element! This will be demonstrated in Section 7.2. ↓

Element Size (mm)	Number of Elements	Output Displacement (mm)
10	32	52.010
default	34	52.016
6	53	52.028
4	79	52.032
3	104	52.033
2	157	52.034
1	312	52.035
0.5	624	52.035



[4] At the end of this study, **Details of Mesh** looks like this. We will use this mesh for another exercise in Section 8.1. ↓

Wrap Up

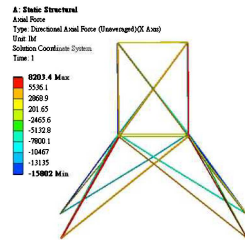
[5] Save the project and exit Workbench. #

Reference

1. Chao-Chieh Lan and Yung-Jen Cheng, 2008, "Distributed Shape Optimization of Compliant Mechanisms Using Intrinsic Functions," *ASME Journal of Mechanical Design*, Vol. 130, 072304.

Section 7.2

3D Truss



7.2.1 About the 3D Truss

[1] Traditionally, a truss is defined as a structure consisting of two-force members; i.e., the members are pin-jointed at ends and the loads are applied on the joints so that the members are either stretched or compressed but not bent. Two members connected by a pin-joint can rotate about the joint independently. In the real world, structural members are rarely connected by pin-joints. Modern structures are constructed using either welds or multiple bolt-and-nuts; i.e., the members are rigid jointed, not pin-jointed. Even in the old days, pin-jointed structures were not common. The main reason for pin-joint assumption was to ease the computational difficulty, in the days when computers were not widespread. Note that, due to the neglect of joint rigidity, pin-joint assumption leads to a conservative design: safer, but over-designed.

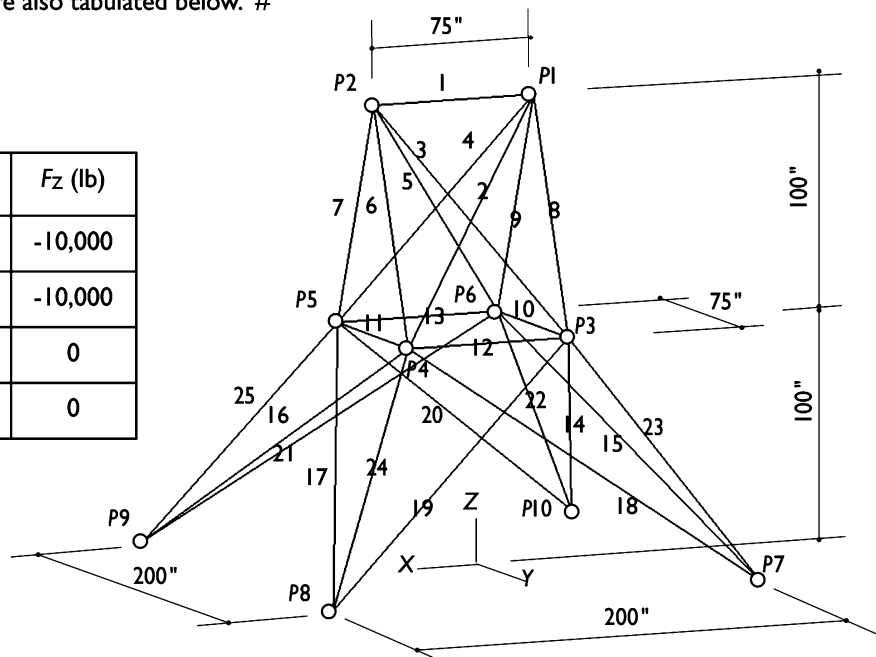
How much is the error caused by the pin-joint assumption? This is a good exercise problem for engineering students (7.4.2, page 310). The amount of error depends on the slenderness of the structural member. If the members are slender enough, there are no essential errors caused by the pin-joint assumption. On the other hand, if the members are not slender enough, then the pin-joints assumption may induce substantial errors.

The beam element **BEAM188**^[Ref 1] (1.3.3[12-13], page 39) is the only element supported in Workbench for line bodies. The so called "truss elements" (such as **LINK180**^[Ref 2]) are not supported in Workbench. To model a pin-jointed structure in Workbench, you need to either specify revolute joints between the members or utilize **End Release** feature (7.4.2, page 310).

In this section, we will create a line model for a power transmission tower as shown below. All members are made of structural steel angle of $1\frac{1}{2} \times 1\frac{1}{2} \times \frac{1}{4}$ cross section. Note that we've assigned a number for each joint (P1-P10) and each member (1-25). The design loads are also tabulated below. #

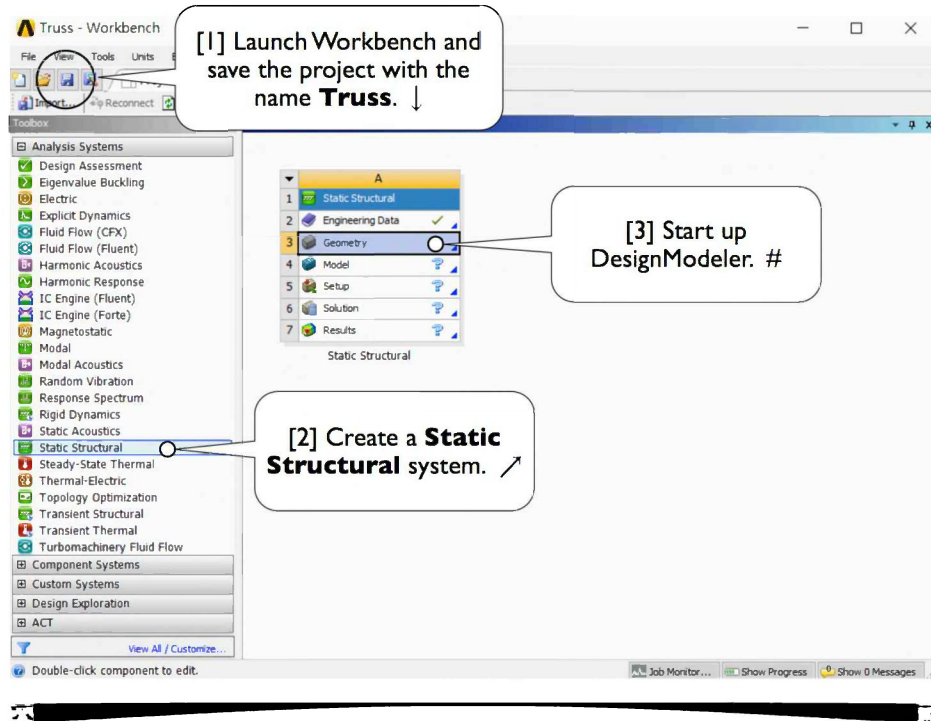
Design Loads

Joint	F_x (lb)	F_y (lb)	F_z (lb)
P1	1,000	-10,000	-10,000
P2	0	-10,000	-10,000
P3	500	0	0
P6	600	0	0

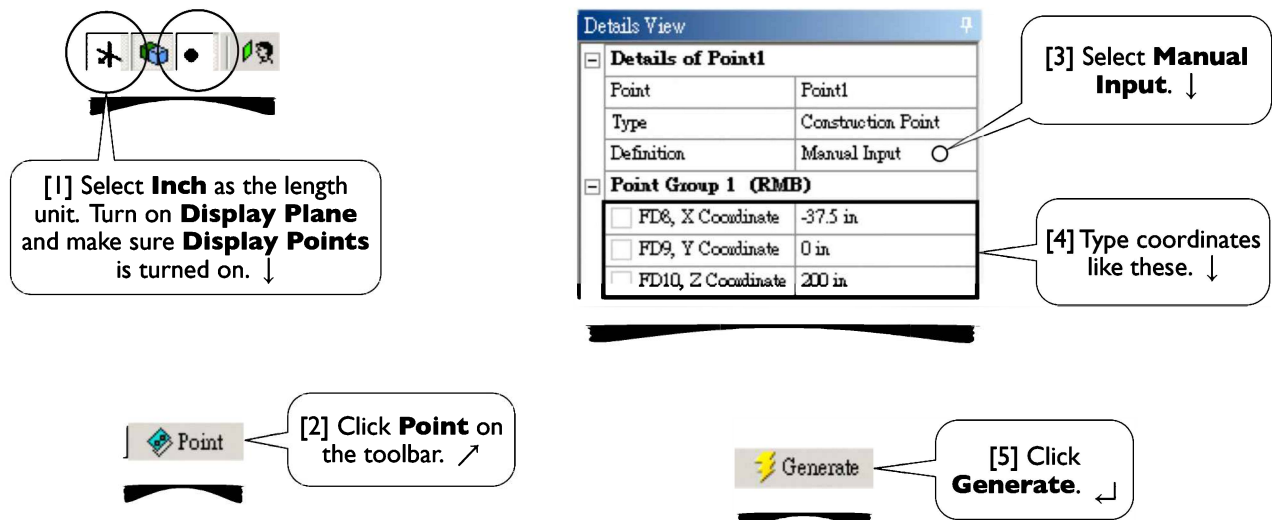


PART A. GEOMETRIC MODELING

7.2.2 Start Up



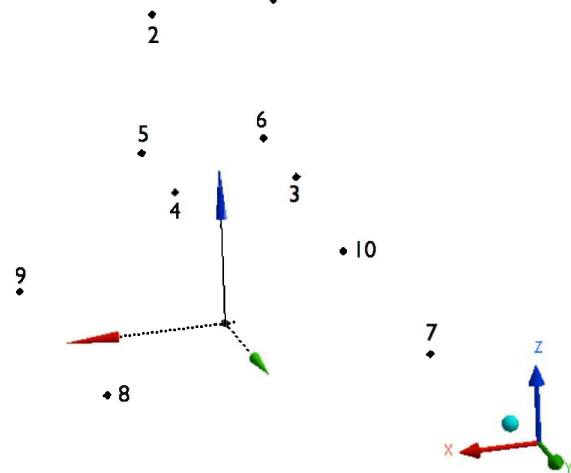
7.2.3 Create 10 Construction Points in 3D Space



Point	X Coordinate (in)	Y Coordinate (in)	Z Coordinate (in)
1	-37.5	0	200
2	37.5	0	200
3	-37.5	37.5	100
4	37.5	37.5	100
5	37.5	-37.5	100
6	-37.5	-37.5	100
7	-100	100	0
8	100	100	0
9	100	-100	0
10	-100	-100	0

[6] Repeat steps [2-5] for additional 9 points (Points 2-10), their coordinates shown here. →

[7] The 10 points. (The numbers are added by the author for your convenience.) Note that the view is rotated such that Z-axis directs upward. ↓



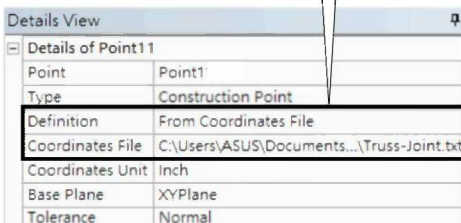
Using **Coordinates File** to Define Points

[8] An alternative way of defining points is to select **From Coordinates File** for **Definition** and read the coordinates from a file [9-10]. When the number of points is large, this is obviously a better way to input coordinates.

A **Coordinates file** [10] is a text file describing the coordinates of points. The file has 5 columns, or fields: (a) group number, (b) ID number, (c) X-coordinate, (d) Y-coordinate, and (e) Z-coordinate. The group number and the ID number can be arbitrarily chosen and they together uniquely identify a point from others.

The fields can be separated by spaces or TABs. You may prepare the text file using a text editor; another way is using a spread-sheet program such as Microsoft Excel and saving as a text file (e.g., a CSV file). ✓

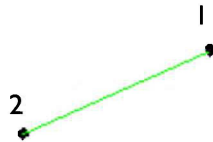
[9] An alternative way of defining coordinates is to select **From Coordinates File** (default) for **Definition** and read the coordinates from a file. →



[10] The format of a **Coordinate File**. #

1	1	-37.5	0	200
1	2	37.5	0	200
1	3	-37.5	37.5	100
1	4	37.5	37.5	100
1	5	37.5	-37.5	100
1	6	-37.5	-37.5	100
1	7	-100	100	0
1	8	100	100	0
1	9	100	-100	0
1	10	-100	-100	0

7.2.4 Create Line Body



[2] Click **Point 1** and control-click **Point 2** to create a line segment (see the joint numbers in [7]). ↓

Member	Start Point	End Point
1	P1	P2
2	P1	P4
3	P2	P3
4	P1	P5
5	P2	P6
6	P2	P4
7	P2	P5
8	P1	P3
9	P1	P6
10	P3	P6
11	P4	P5
12	P3	P4
13	P5	P6
14	P3	P10
15	P6	P7
16	P4	P9
17	P5	P8
18	P4	P7
19	P3	P8
20	P5	P10
21	P6	P9
22	P6	P10
23	P3	P7
24	P4	P8
25	P5	P9

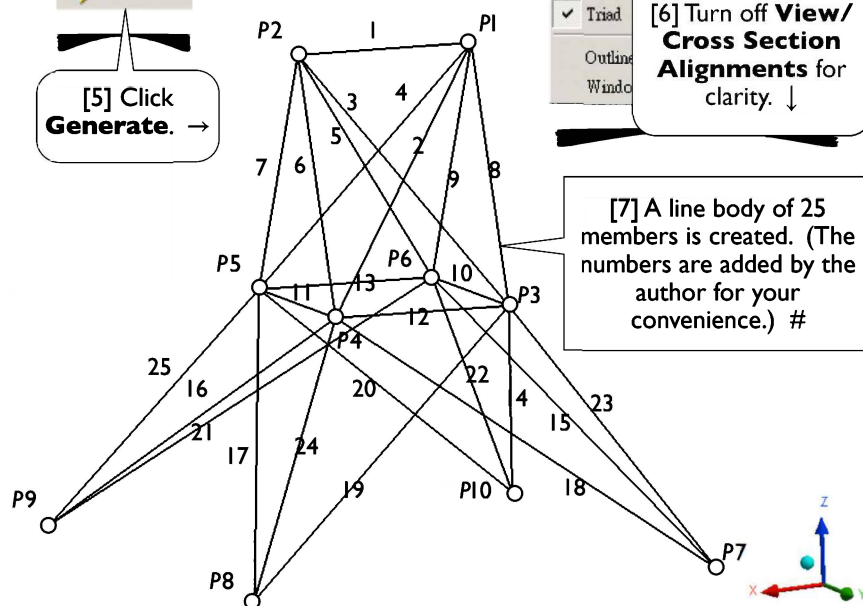
[3] Repeat step [2] for additional 24 line segments (25 line segments in total). Each line segment is created by clicking the starting point and then control-clicking the ending point. If you make a mistake, you can remove a line segment by clicking its starting point and then control-clicking ending point again. When creating or removing a line segment, order of starting/ending points is not relevant. ↓

Details View	
Details of Line1	
Lines From Points	Line1
Point Segments	25
Operation	Add Material

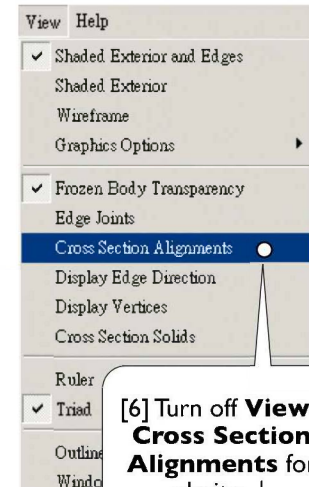
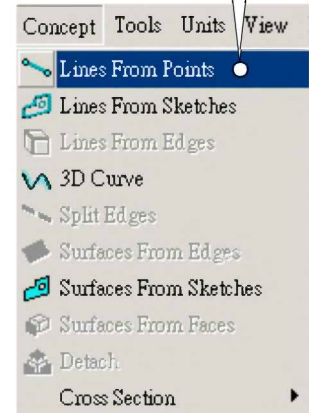
[4] Click **Apply**. ✓

Generate

[5] Click **Generate**. →



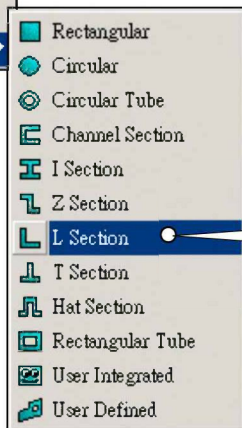
[1] Pull-down-select **Concept/Lines From Points**. ←



[6] Turn off **View/Cross Section Alignments** for clarity. ↓

[7] A line body of 25 members is created. (The numbers are added by the author for your convenience.) #

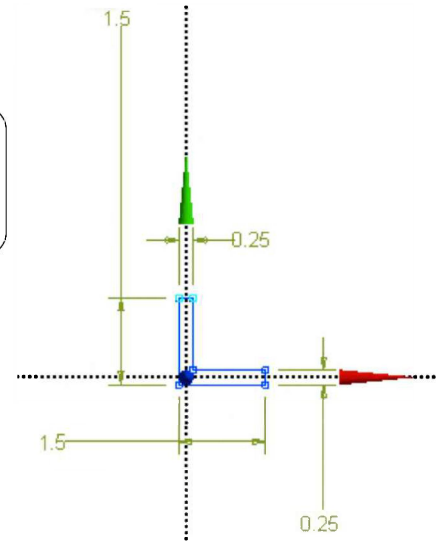
7.2.5 Create Cross Section



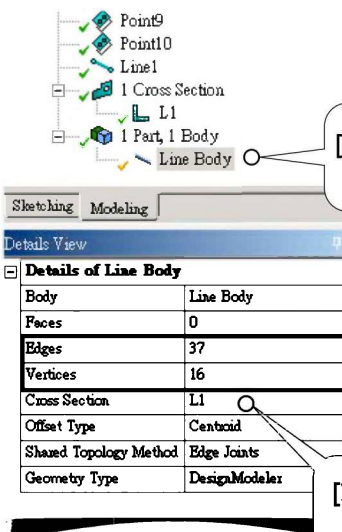
Details View	
Details of L1	
Sketch	L1
Show Constraints?	No
Dimensions: 4	
W1	1.5 in
W2	1.5 in
t1	0.25 in
t2	0.25 in

[2] Type dimensions for the steel angle. You don't have to click **Generate**. #

[1] Pull-down-select **Concept/Cross Section/L Section**. ↑



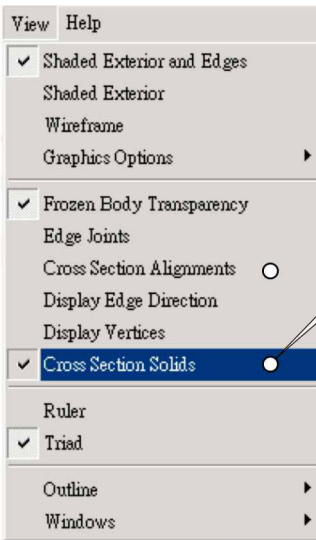
7.2.6 Assign Cross Section to the Line Body



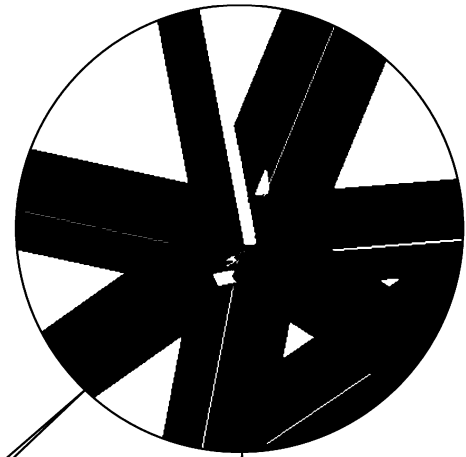
[1] Select **Line Body**. ↓

[2] Select the cross section **L1**. →

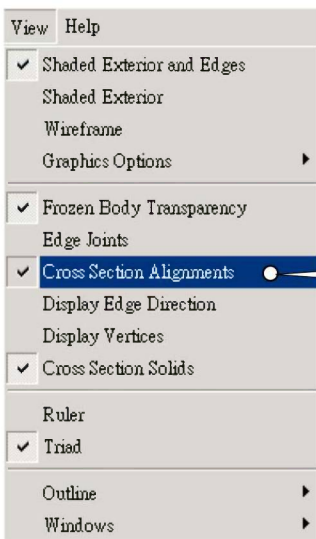
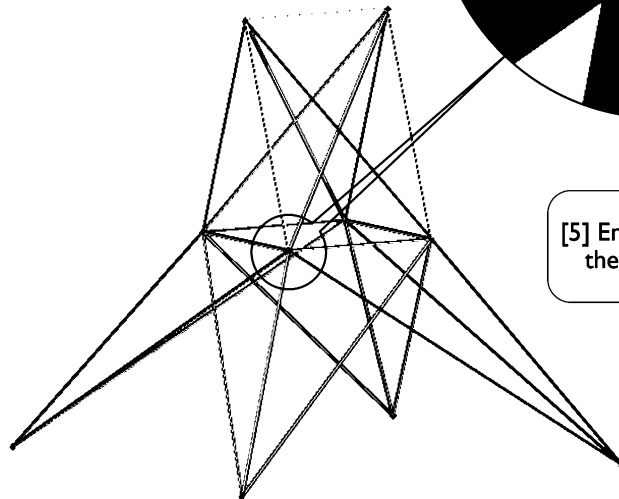
[3] Although we created only 10 joints and 25 members, the details view shows that there are 16 vertices and 37 edges. An additional 6 vertices and 12 edges are created because 6 pairs of members cross over each other and are treated as if they are rigidly welded to each other at points of cross-over. We assume that this is the real situation and accept this default arrangement. ↩



[4] Turn on **View/Cross Section Solids** and make sure **Cross Section Alignments** is off. ↘

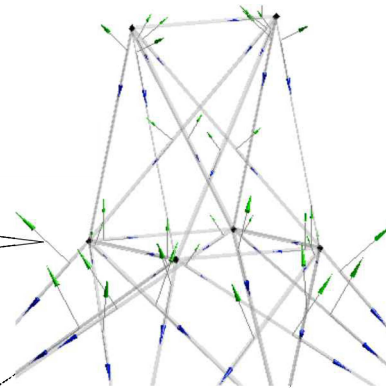


[5] Enlarge to view the details. ✓



[6] Turn on **View/Cross Section Alignments**. ↓

[7] The default cross section alignments. Close DesignModeler. ↓

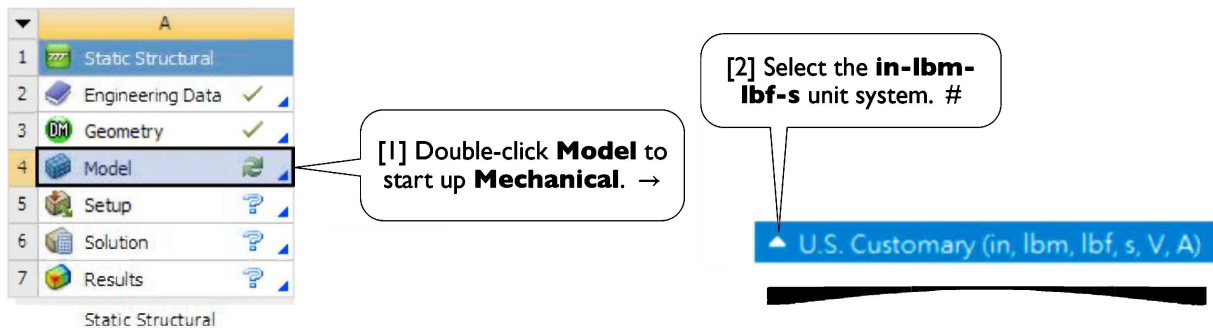


Cross Section Alignments

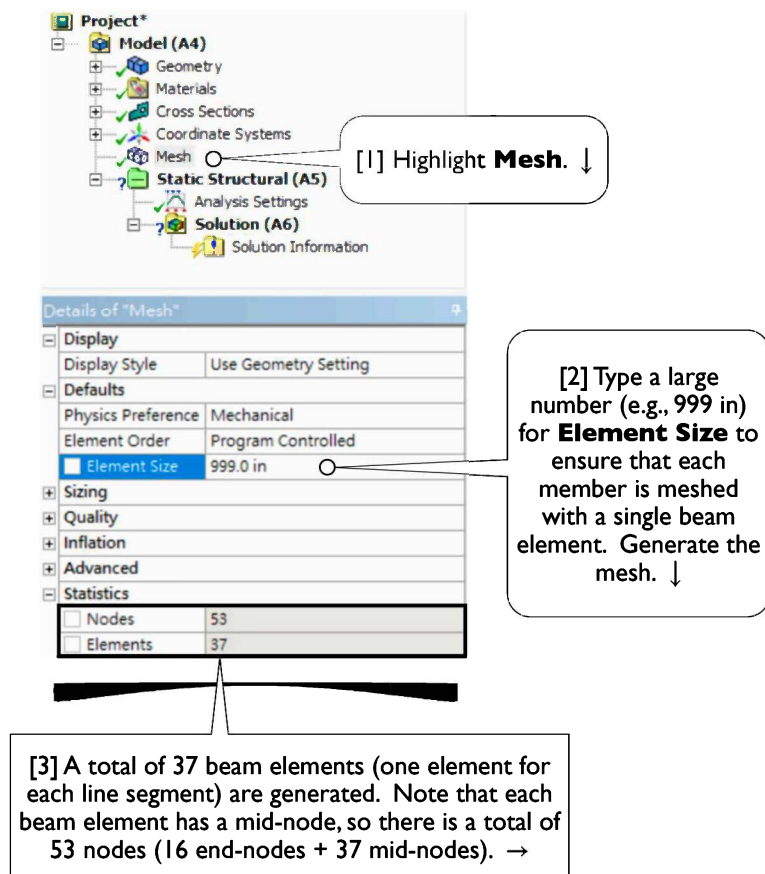
[8] The default cross section alignments [7] are not entirely consistent with the real situation in this case. However, we decided to neglect the difference between the reality and the default alignments. Since the structural members are quite slender, the behaviors are essentially two-force members; therefore, alignments should not be critical. In other words, cross section alignments have little effect on the structural response in this case. We will demonstrate the adjustment of cross section alignments in 7.3.9 (page 300), in which the alignments must be adjusted, otherwise it would deviate from the reality too much. #

PART B. SIMULATION

7.2.7 Start Up **Mechanical**



7.2.8 Generate Mesh



Why mesh each member with a single element?

[4] We mentioned (7.1.16[1], page 282) that we can obtain a solution equal to analytical values by meshing each straight beam with a single element. The reason we meshed each member with a single beam element here is to demonstrate this fact.

The default settings of **Mesh** would mesh the model with 205 beam elements and would result in exactly the same solution as 37 elements. As an exercise (7.4.2, page 310), verify it yourself after the completion of this section.

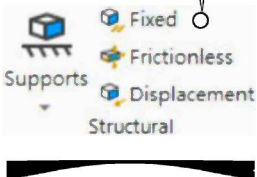
Use Surface/Line Models

Whenever Suitable

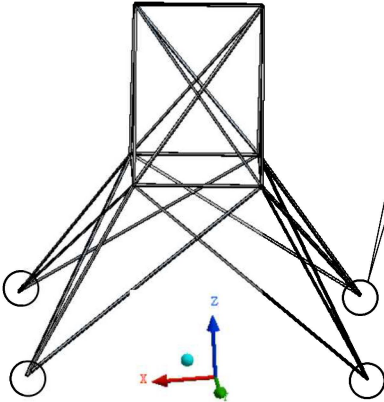
Since the solution of a model meshed with beam elements or shell elements converges very fast (i.e., very accurate solution can be obtained with only a few elements), we should consider a line model or surface model whenever suitable. This is particularly true for those problems requiring many iterations or substeps, such as nonlinear problems, dynamic problems, optimization problems, etc. #

7.2.9 Specify Supports

[1] Highlight **Static Structural** in the project tree and insert a **Fixed Support**. →



[2] Control-select the 4 vertices at the base. You may need to turn on vertex select filter. ↓

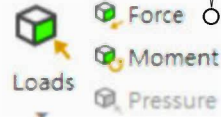


[3] And click **Apply**. #

Details of "Fixed Support"	
Scope	
Scoping Method	Geometry Selection
Geometry	4 Vertices
Definition	
Type	Fixed Support
Suppressed	No

7.2.10 Specify Loads

[1, 3, 5, 7] Click **Force**.
→ → ↓ ↘



[2] Select P1 (see figure below) and click **Apply**. ←

Details of "Force"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	1000 lbf (ramped)
Y Component	-10000 lbf (ramped)
Z Component	-10000 lbf (ramped)
Suppressed	No

[4] Select P2 and click **Apply**. ←

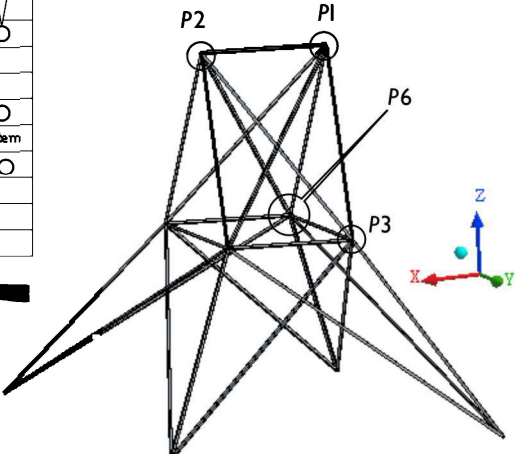
Details of "Force 2"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	0 lbf (ramped)
Y Component	-10000 lbf (ramped)
Z Component	-10000 lbf (ramped)
Suppressed	No

[6] Select P3 and click **Apply**. ↑

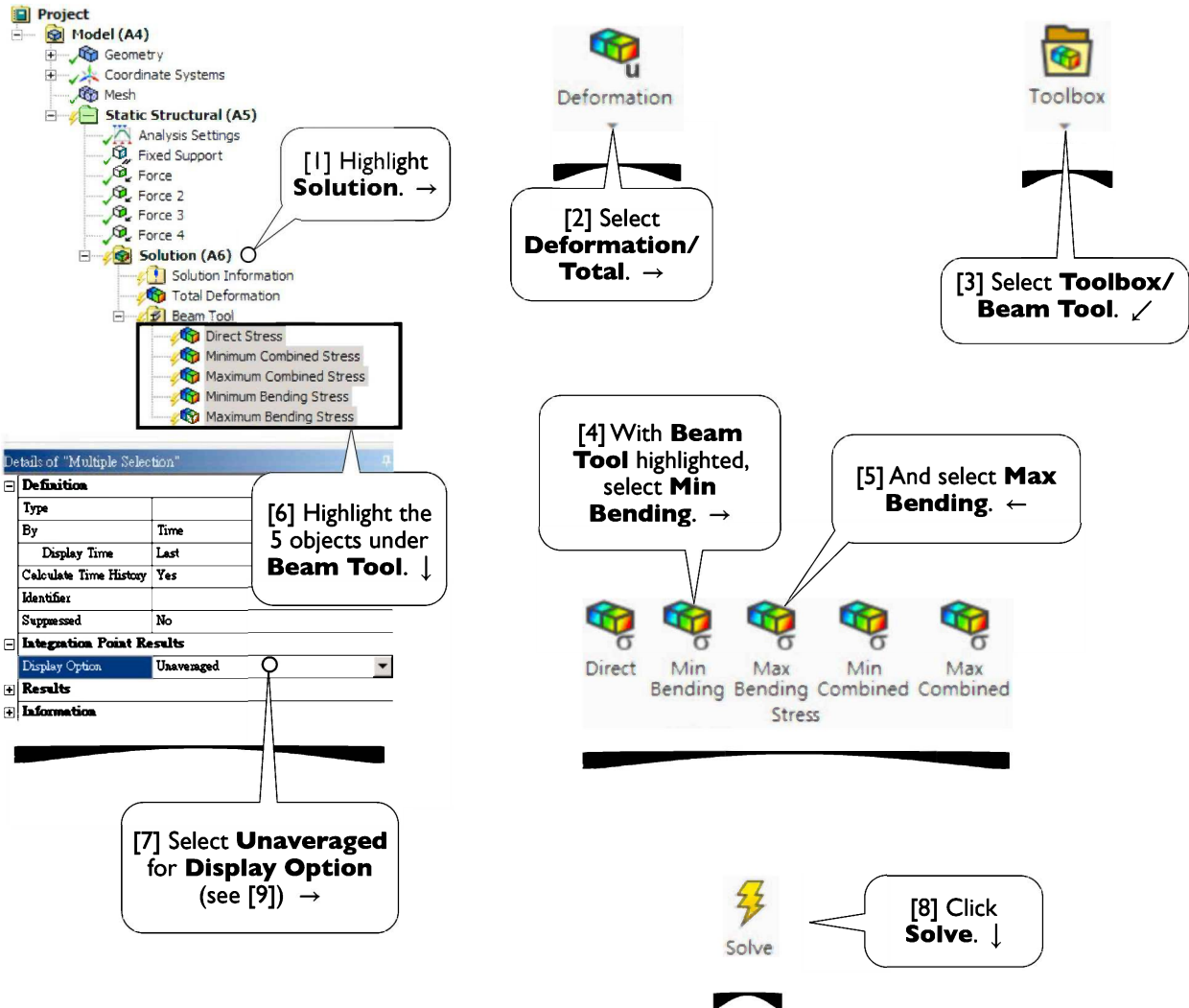
Details of "Force 3"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	500 lbf (ramped)
Y Component	0 lbf (ramped)
Z Component	0 lbf (ramped)
Suppressed	No

[8] Select P6 and click **Apply**. #

Details of "Force 4"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	600 lbf (ramped)
Y Component	0 lbf (ramped)
Z Component	0 lbf (ramped)
Suppressed	No



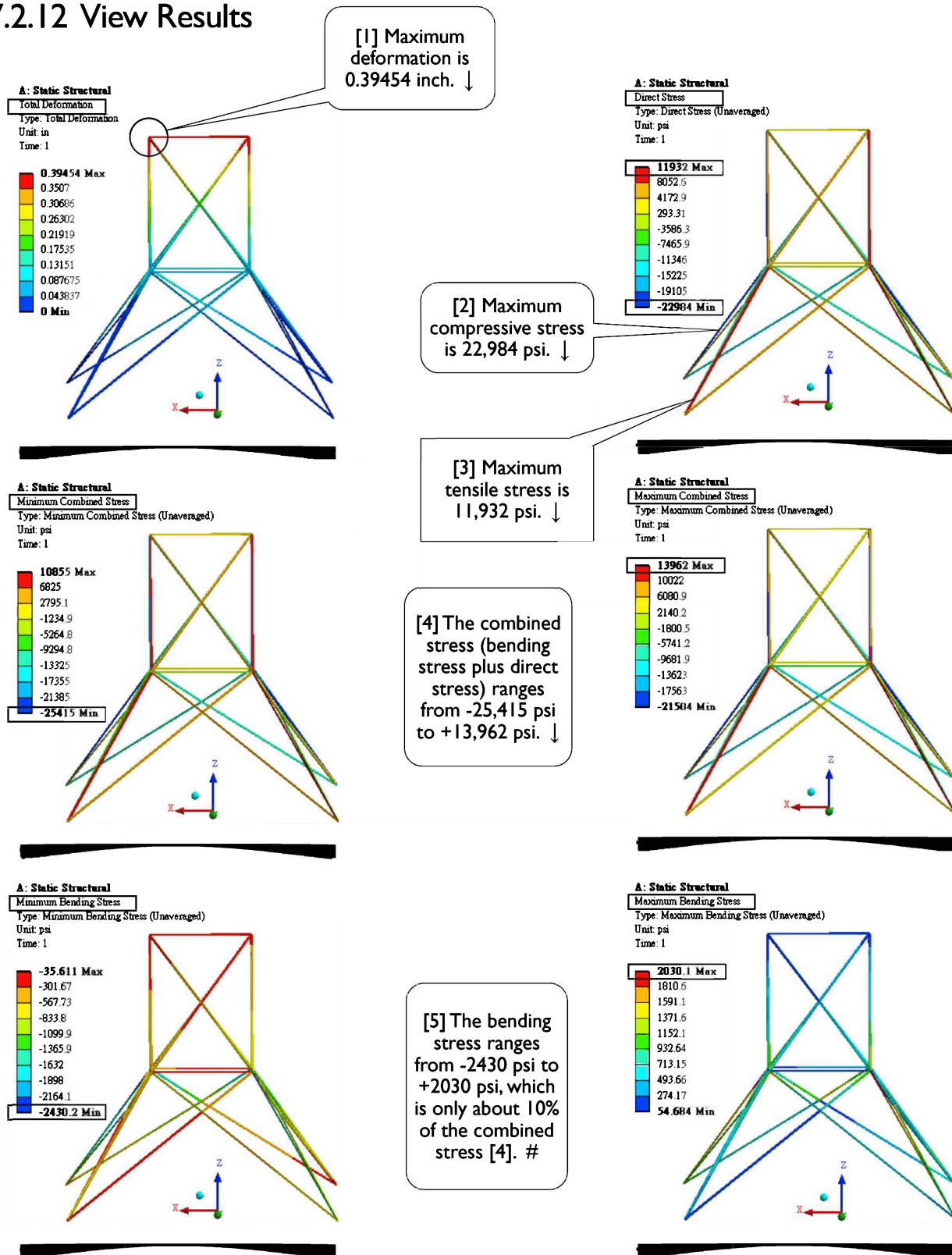
7.2.11 Set Up Solution Branch and Solve the Model



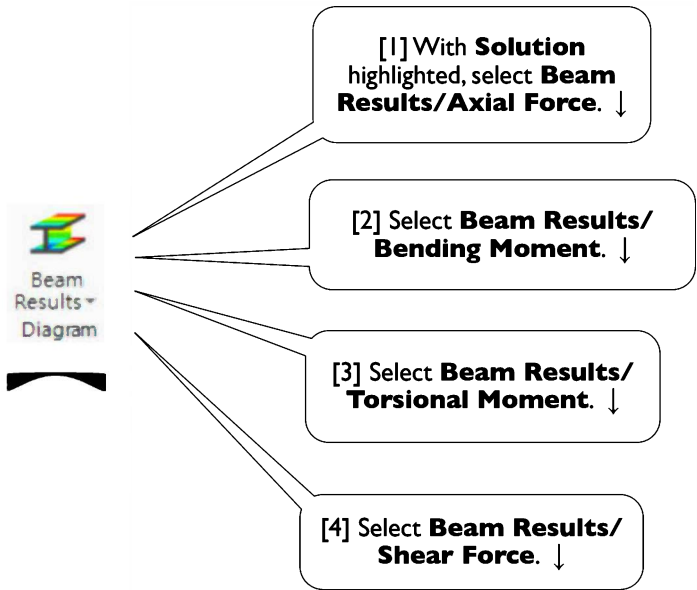
Why Unaveraged Stresses?

[9] We've introduced the notions of averaged and unaveraged stresses (3.5.6, page 160). Unaveraged stresses are also called *element stresses*, since they are calculated at points (usually the geometric center or the integration points) inside elements. On the other hand, averaged stresses are also called *nodal stresses*, since they are calculated at nodes, which are located at element boundaries. Since we mesh each member with a single beam element, and if we select to display averaged stresses, every two adjacent members' stresses would have been averaged and reported. The averaged stresses in this case would not have any meaning. #

7.2.12 View Results



7.2.13 View Member Forces/Moments

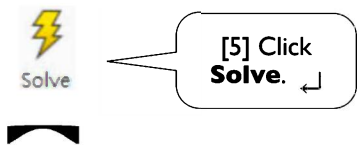


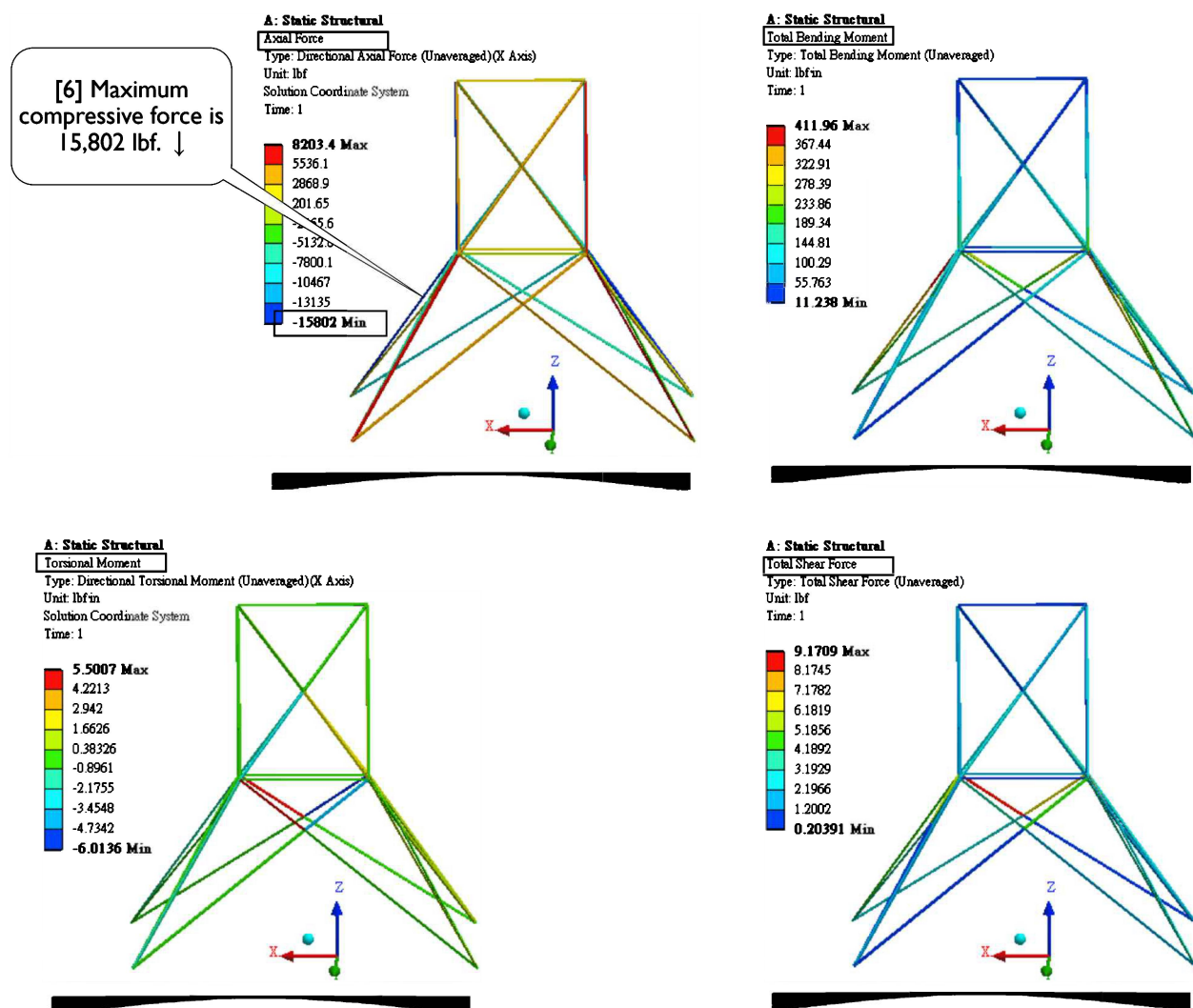
Details of "Axial Force"	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	All Line Bodies
[-] Definition	
Type	Directional Axial Force
By	Time
Display Time	Last
Coordinate System	Solution Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
[-] Integration Point Results	
Display Option	Unaveraged
+ Results	
+ Minimum Value Over Time	
+ Maximum Value Over Time	
+ Information	

Details of "Total Bending Moment"	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	All Line Bodies
[-] Definition	
Type	Total Bending Moment
By	Time
<input type="checkbox"/> Display Time	Last
Calculate Time History	Yes
Identifier	
Suppressed	No
[-] Integration Point Results	
Display Option	Unaveraged
+ Results	
+ Minimum Value Over Time	
+ Maximum Value Over Time	
+ Information	

Details of "Torsional Moment"	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	All Line Bodies
[-] Definition	
Type	Directional Torsional Mo...
By	Time
Display Time	Last
Coordinate System	Solution Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
[-] Integration Point Results	
Display Option	Unaveraged
+ Results	
+ Minimum Value Over Time	
+ Maximum Value Over Time	
+ Information	

Details of "Total Shear Force"	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	All Line Bodies
[-] Definition	
Type	Total Shear Force
By	Time
<input type="checkbox"/> Display Time	Last
Calculate Time History	Yes
Identifier	
Suppressed	No
[-] Integration Point Results	
Display Option	Unaveraged
+ Results	
+ Minimum Value Over Time	
+ Maximum Value Over Time	
+ Information	





Wrap Up

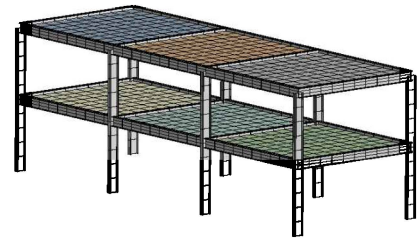
[7] Save the project and exit Workbench. #

References

1. All Help>Mechanical APDL>Theory Reference>13.188. BEAM188
2. All Help>Mechanical APDL>Theory Reference>13.180. LINK180

Section 7.3

Two-Story Building

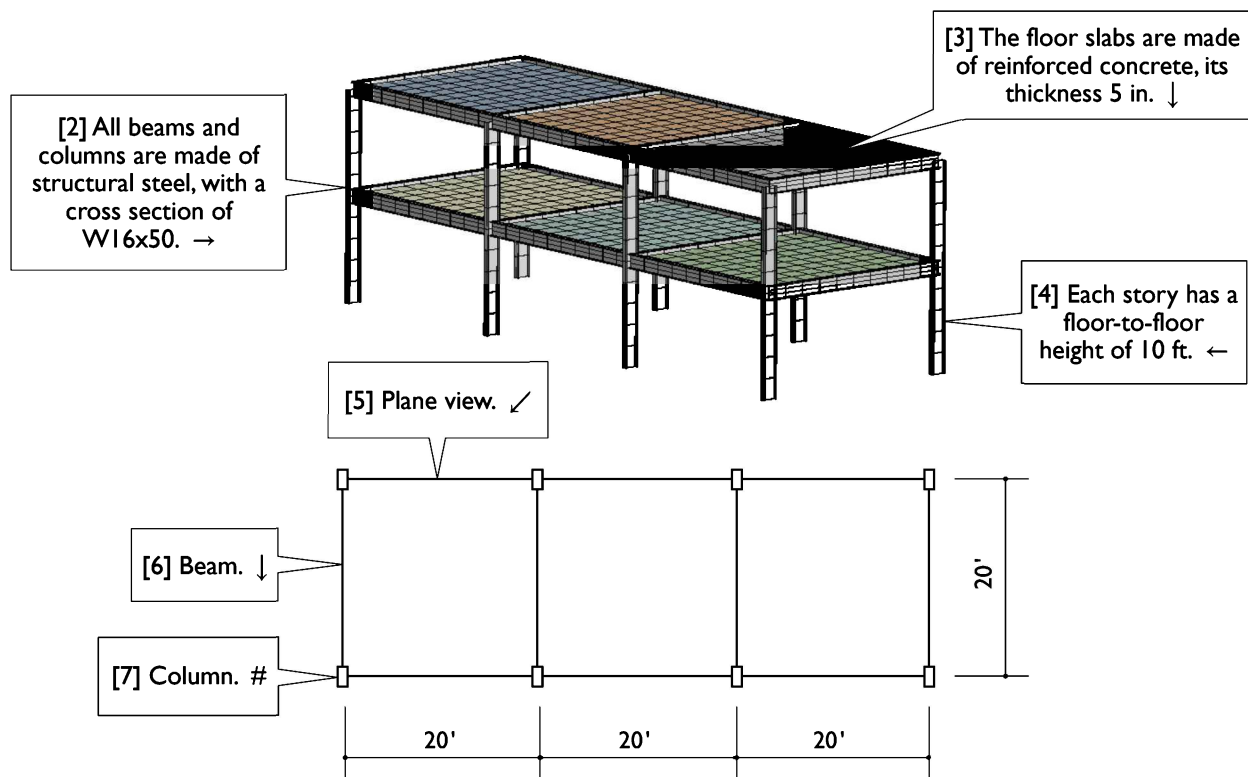


7.3.1 About the Two-Story Building

[1] The two-story building [2-7] is constructed for residential usage. The local building code requires that a live load of 50 lb/ft² be considered, along with its own weight. Since the building is in an earthquake zone, an earthquake load must be considered. For a low-rise building like this, the building code allows an equivalent static analysis instead of a dynamic analysis. Here we consider a static earthquake load, which is equivalent to 0.2 times gravitational acceleration, applying horizontally in the shorter direction of the building. In a practical design project, earthquake load applying in other directions should also be simulated.

The beams and columns will be modeled as line bodies and the floor slabs as surface bodies. We assume that all bodies are perfectly bonded together. The geometric model will be used for a static structural simulation in this section. The model will be used again in Section 11.2 for a modal analysis and Section 12.3 for a harmonic response analysis.

In this exercise, we will use **in-lbm-lbf-s** unit system most of the time. On some occasions, we will change the unit system to **ft-lbm-lbf-s** unit system. One feature of Workbench is the flexibility of using unit systems. When you switch to another unit system, Workbench will convert the units nicely for you. ✓



PART A. GEOMETRIC MODELING

7.3.2 Start Up

[1] Launch Workbench.
Create a **Static Structural** system. Save the project as **Building**. →



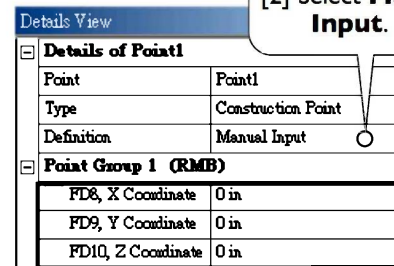
[2] Start up DesignModeler. #

7.3.3 Create 16 Points in the Space

Point	X Coordinate (in)	Y Coordinate (in)	Z Coordinate (in)
1	0	0	0
2	240	0	0
3	480	0	0
4	720	0	0
5	0	0	240
6	240	0	240
7	480	0	240
8	720	0	240
9	0	120	0
10	240	120	0
11	480	120	0
12	720	120	0
13	0	120	240
14	240	120	240
15	480	120	240
16	720	120	240



[1] Select **Inch** as the length unit. Click **Point** on the toolbar. ↓



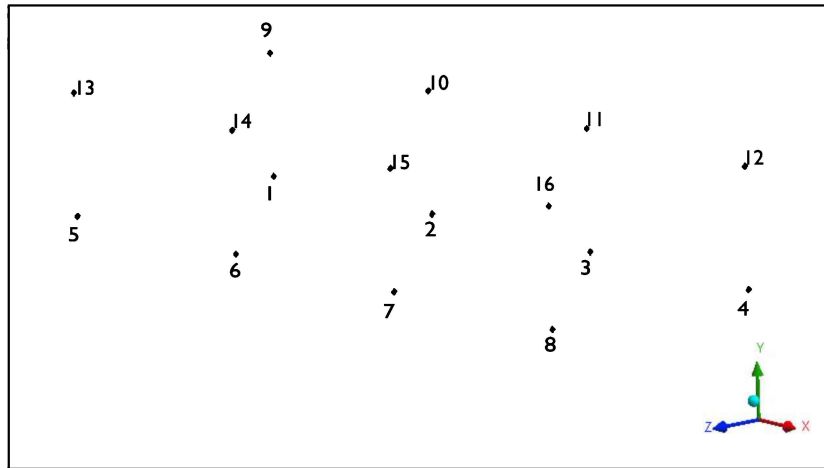
[2] Select **Manual Input**. ↓

[3] Type coordinates for **Point1**. ↓



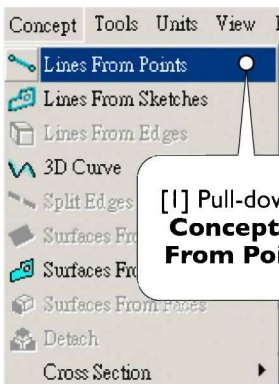
[4] Click **Generate**. ↓

[5] Repeat steps [1-4] for additional 15 points (Points 2-16) and type their respective coordinates shown in this table. Remember that, alternatively, you may input the coordinates by preparing a coordinates file (see 7.2.3[8-10], page 285). ↩



[6] These 16 points are starting and ending points of the 8 columns for the lower floor. (The numbers are added by the author for your convenience.) #

7.3.4 Create a Line Body of 10 Beams

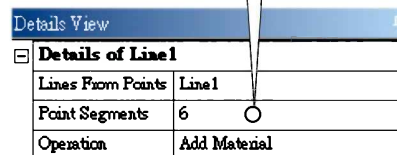


[1] Pull-down-select **Concept/Lines From Points.** →

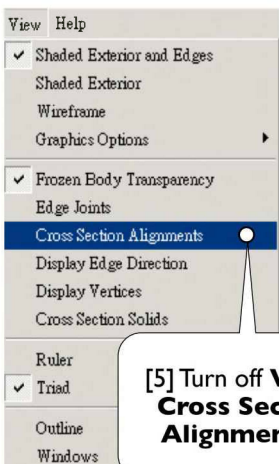
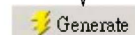
[2] Define 6 line segments. Remember that each line segment is defined by first clicking its starting point and then control-clicking its ending point. ↓

Member	Start Point	End Point
1	P9	P12
2	P13	P16
3	P9	P13
4	P10	P14
5	P11	P15
6	P12	P16

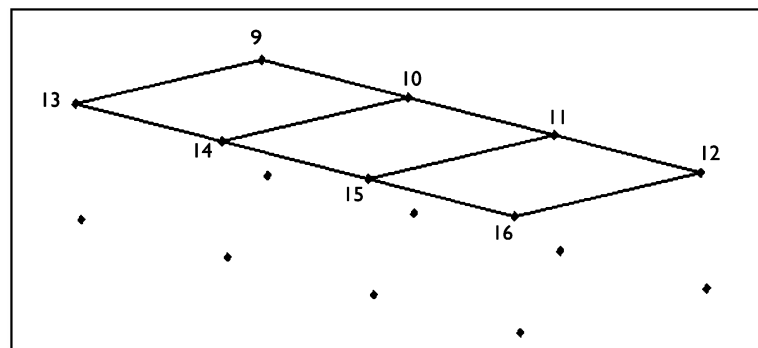
[3] Click **Apply.** →



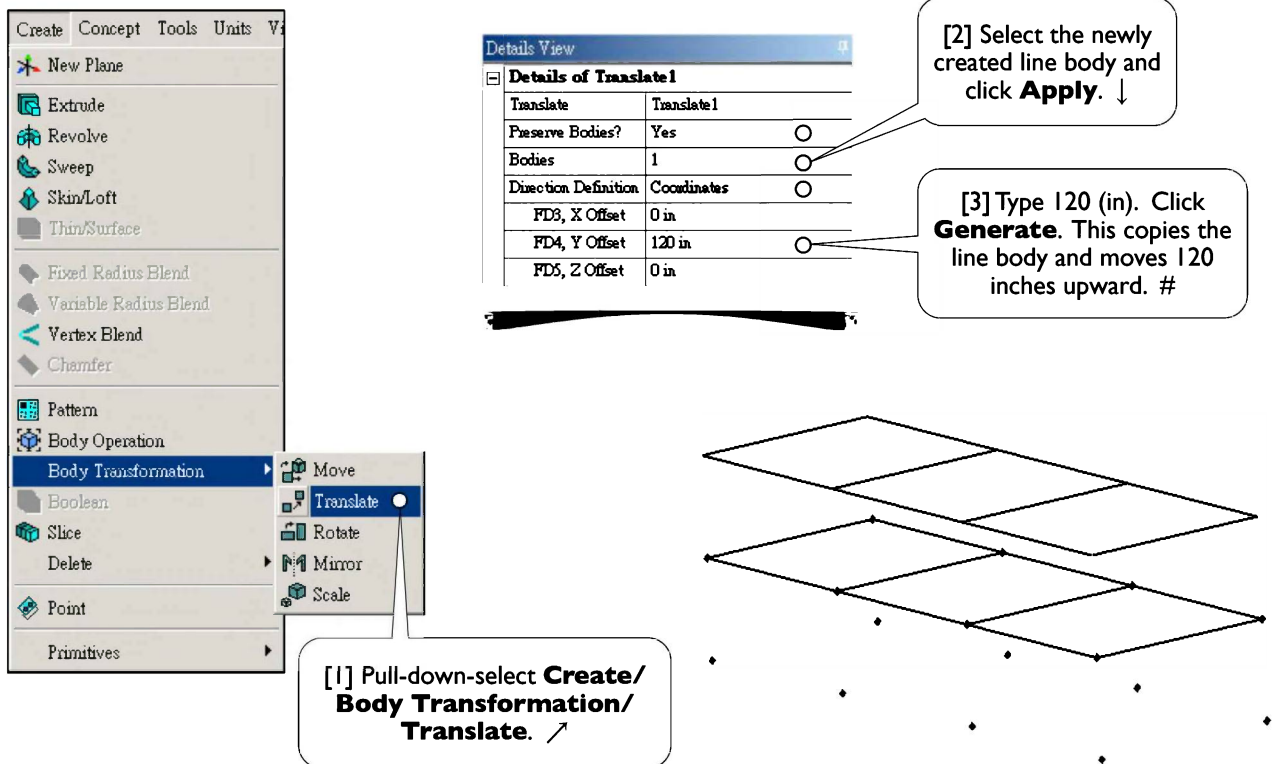
[4] Click **Generate.** ✓



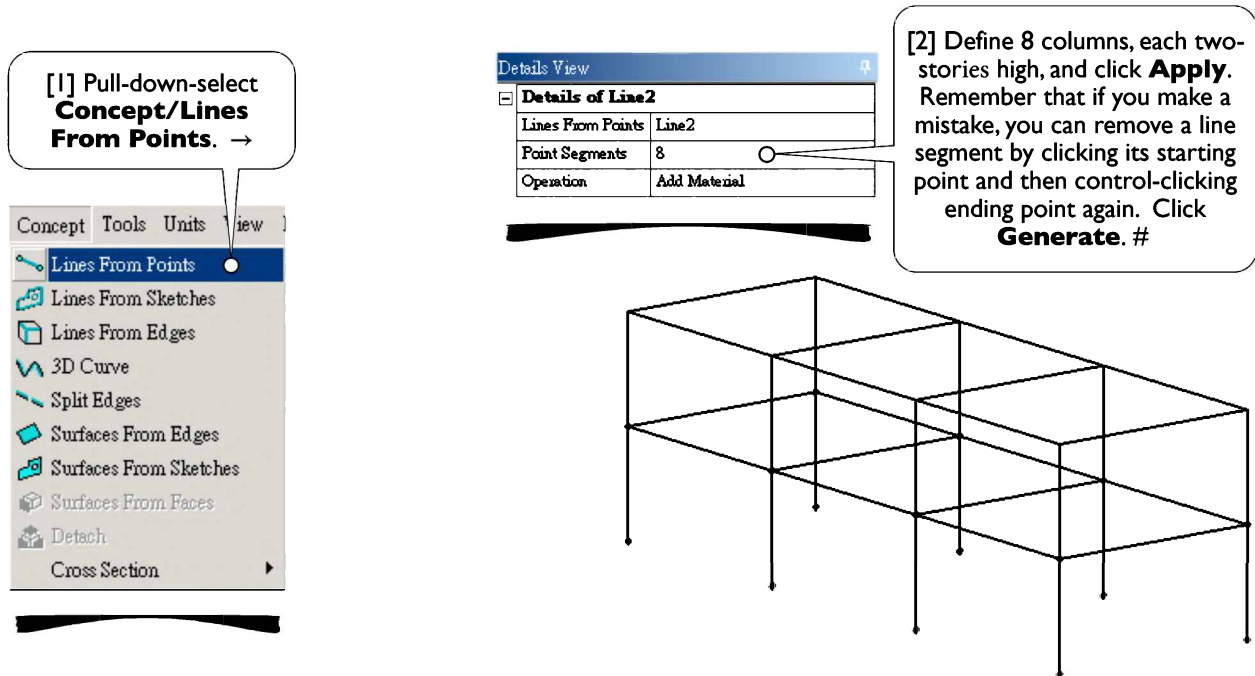
[5] Turn off **View/Cross Section Alignment.** #



7.3.5 Copy the Line Body



7.3.6 Create Line Body for Columns



7.3.7 Create a Cross Section

[1] Pull-down-select **Concept/Cross Section/I Section.** →

[2] Type the dimensions of the W16x50 cross section like this. #

Details View	
Details of I1	
Sketch	I1
Show Constraints?	No
Dimensions: 6	
W1	7.07 in.
W2	7.07 in.
W3	16.25 in.
t1	0.63 in.
t2	0.63 in.
t3	0.38 in.

The diagram shows an I-section cross-section with the following dimensions: W1 (flange width) = 7.07, W2 (web width) = 7.07, W3 (total height) = 16.25, t1 (flange thickness) = 0.63, t2 (web thickness) = 0.63, and t3 (fillet radius) = 0.38.

7.3.8 Assign the Cross Section to the Line Body

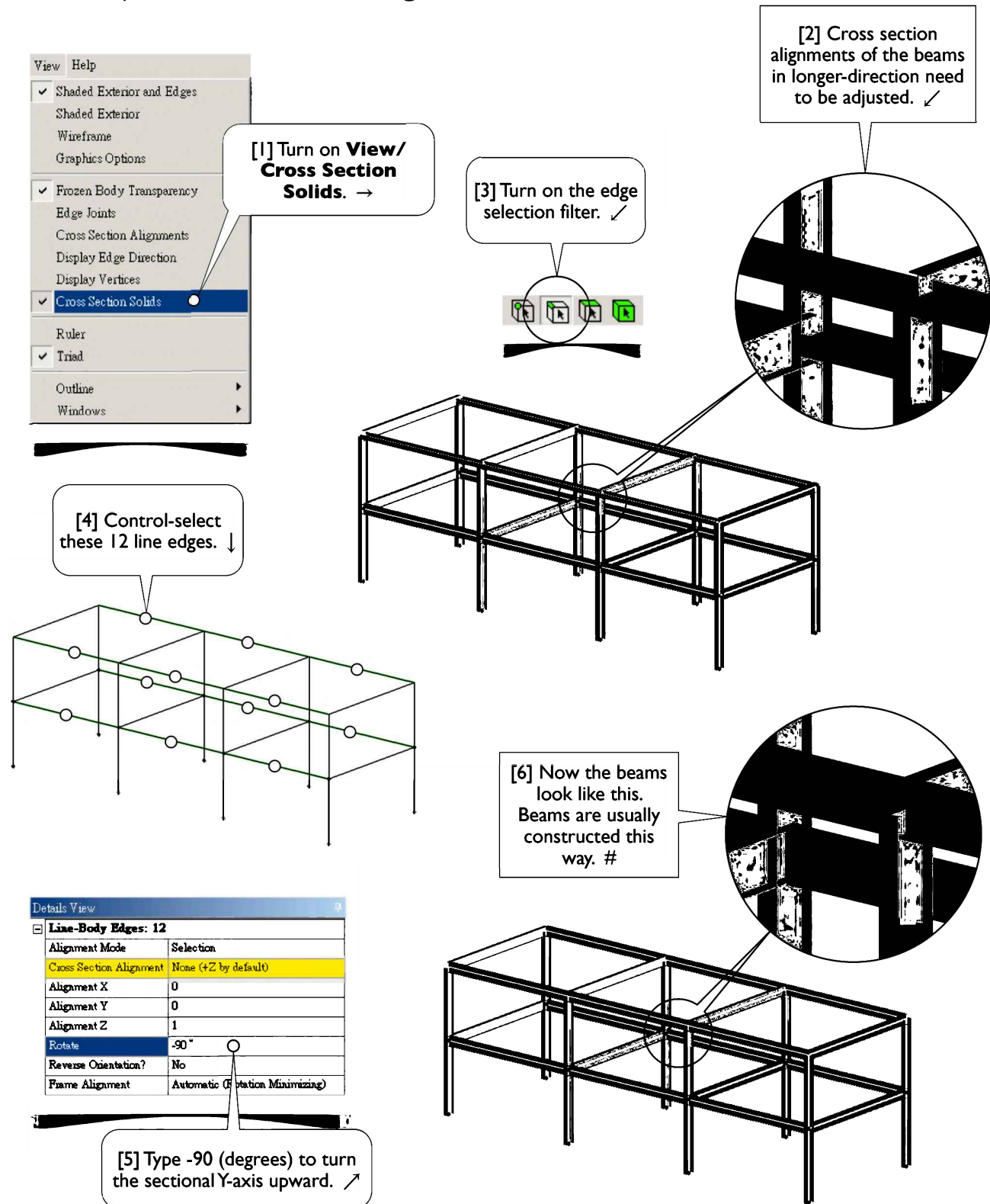
[1] Select **Line Body** in the model tree. ↓

[2] Assign I1 to the line body. #

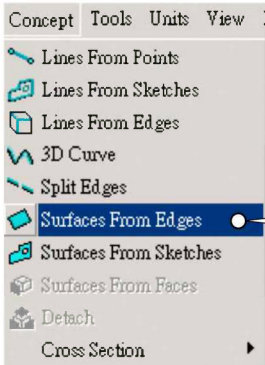
Details View	
Details of Line Body	
Body	Line Body
Faces	0
Edges	36
Vertices	24
Cross Section	I1
Offset Type	Centroid
Shaded Topology Method	Edge Joints
Geometry Type	DesignModeler

The diagram shows a line body with a cross-section assigned. The cross-section is labeled I1, and the line body is labeled Line Body.

7.3.9 Adjust Cross Section Alignments

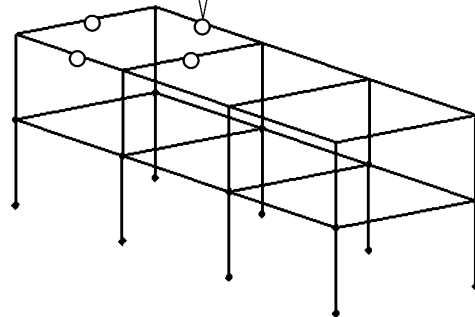


7.3.10 Create Surface Bodies

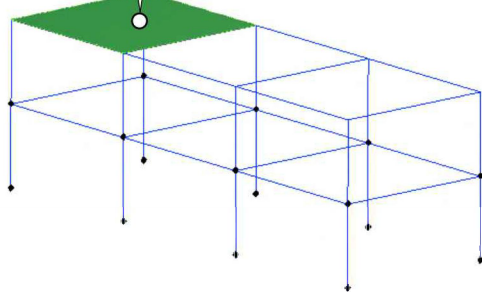


[1, 6] Pull-down-select **Concept/Surfaces From Edges**. → ↓

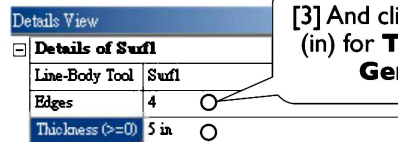
[2] Control-select these 4 edges. Notice that the cursor turns to the line-edge shape. ↓



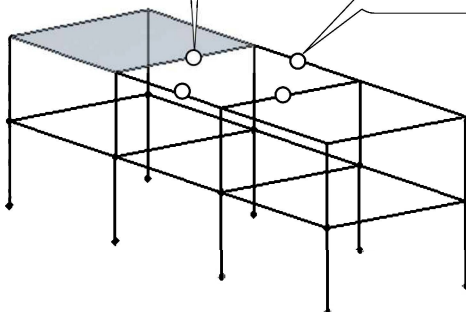
[4] Turn on the face selection filter and click the surface body. One of the surface faces is highlighted (with a green color), indicating that it is the positive-side of the surface body; the other side is the negative-side. ↘



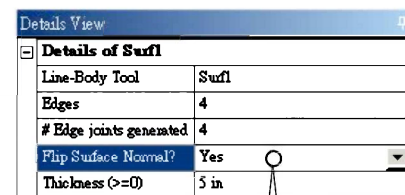
[3] And click **Apply**. Type 5 (in) for **Thickness**. Click **Generate**. ←



[7] Select this edge. Use **Selection Panes** (see [8]) to correctly select a line edge rather than a surface edge. ↘

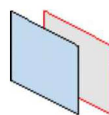


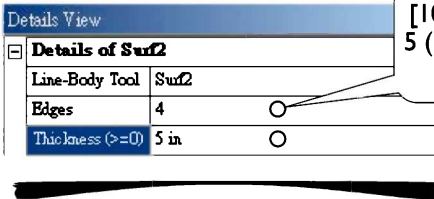
[9] Control-select the other 3 edges. ↙



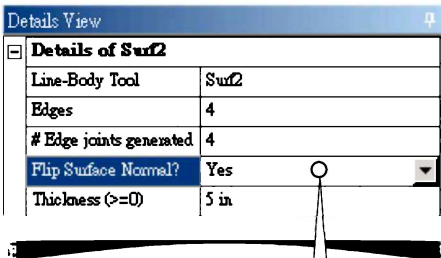
[5] If the positive-side of the surface body is not upward, select **Surf1** in the model tree, and, in the details view, select **Yes** for **Flip Surface Normal?**. Click **Generate** and go back to [4] to make sure the positive-side is upward. ↘

[8] **Selection Panes** can be used to select a line edge (instead of a surface edge). In this case one pane represents a line edge and the other pane represents a surface edge. You can distinguish them by the cursor shape. Also, you can deselect a pane by control-selecting it. ↙



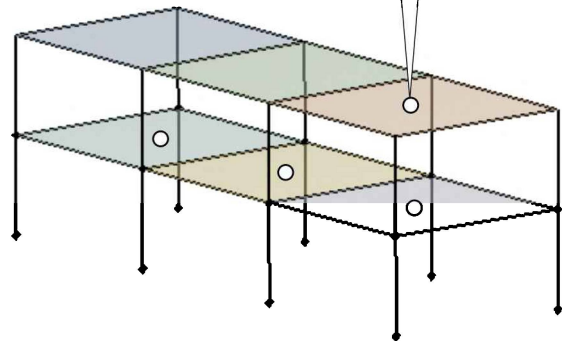


[10] And click **Apply**. Type 5 (in) for **Thickness**. Click **Generate**. ↓



[11] Remember, if the positive-side of the surface body is not upward, select the surface in the model tree, and, in the details view, select **Yes** for **Flip Surface Normal?** (see [4-5], last page). →

[12] Repeat steps [6-11] to create 4 more surface bodies. ✓



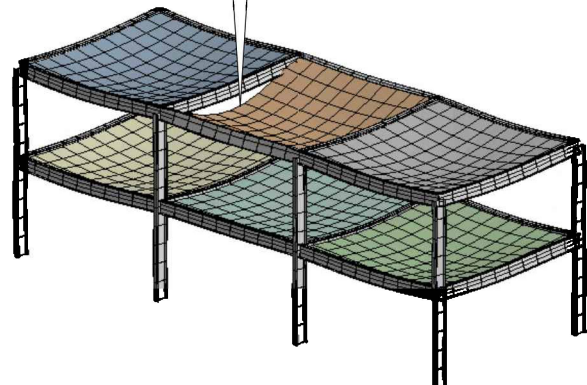
Edge Selection

[13] When you select edges, you may need to use **Selection Panes** to correctly select the line edge rather than a surface edge ([7-8], last page). You can distinguish them by the cursor shape. If you select a surface edge instead of a line edge, the slab would not connect to the line body and would have a gap after deformation [14]. →

[14] If you select a surface edge instead of a line edge, the slab would not connect to the line body and would have a gap after deformation. ←

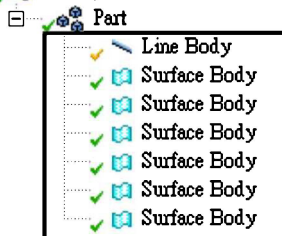
Direction of Surface Bodies

[15] Each surface body has a positive-side and a negative-side. In **Mechanical**, the pressure always applies on the positive-side. Therefore it is important to know which is the positive-side: turn on the face selection filter and click the surface body, and the positive-side will be highlighted with a green color. Make sure the positive-side of each surface body is upward in this case. If it is not, select **Yes** for **Flip Surface Normal?** in the details view (see [4-5], last page). #



7.3.11 Form a Single Part

1 Part, 7 Bodies



[1] Select all the 7 bodies and right-click-select **Form New Part**. This bonds all the bodies together: the deformation will be compatible at the boundaries between bodies. Close DesignModeler. #

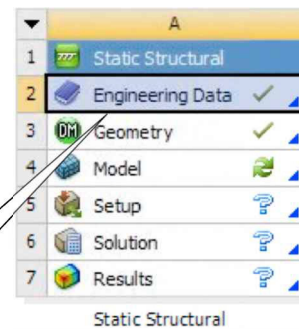
PART B. SIMULATION

7.3.12 Prepare Material Properties

Project

[4] Click to return to **Project Schematic**. #

[1] Double-click **Engineering Data**. ↓



Engineering Data Sources

	A	B	C	D
1	Data Source			
2	Favorites			
3	General Materials			
4	General Non-linear Materials			
5	Explicit Materials			Material samples for use in an explicit analysis.

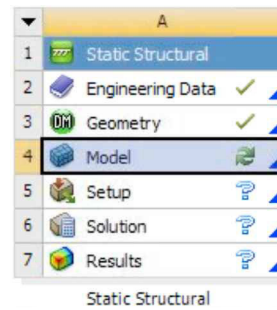
[2] Click **Engineering Data Sources** and click to load the **General Materials** library (also see 6.3.8[2-4], page 262). ↓

Outline of General Materials

	A	B	C	D	E
1	Contents of General Materials		Add	source	Description
2	Material				
3	Air				General properties for air.
4	Aluminum Alloy				General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277.
5	Concrete				
6	Copper Alloy				
7	Gray Cast Iron				
8	Magnesium Alloy				
9	Polyethylene				
10	Silicon Anisotropic				
11	Stainless Steel				
12	Structural Steel				Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5 -110.1
13	Titanium Alloy				

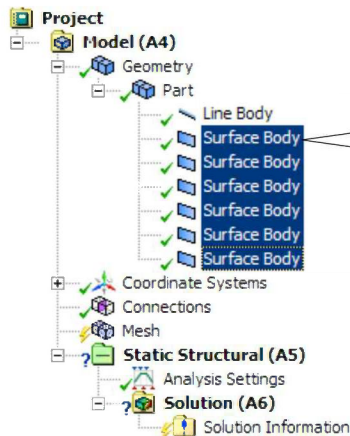
[3] Click the plus sign next to **Concrete** to add it to **Engineering Data**. ↖

7.3.13 Start Up **Mechanical**

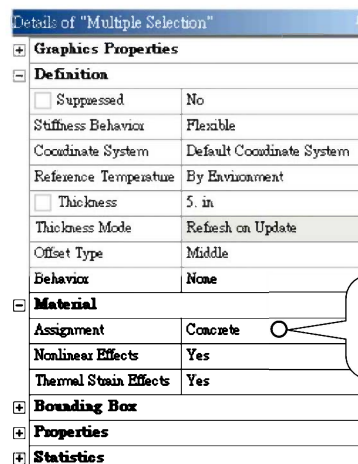


[1] Start up **Mechanical**. Make sure the unit system is **in-lbm-lbf-s**. #

7.3.14 Assign Materials



[1] Highlight all the surface bodies. →

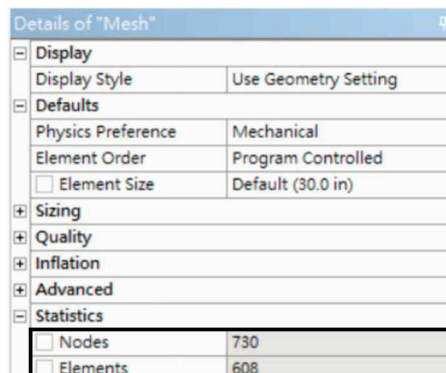


[2] Select **Concrete**. #

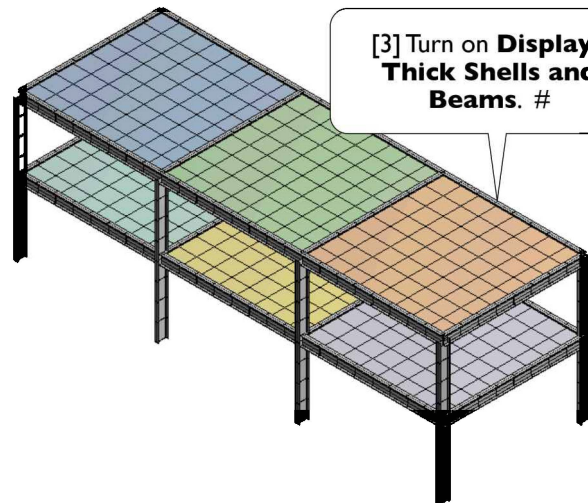
7.3.15 Generate Mesh



[1] Highlight **Mesh** and generate mesh. ✓

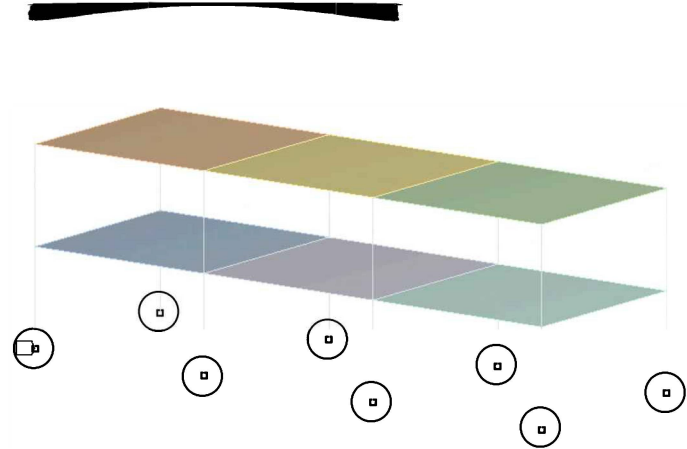
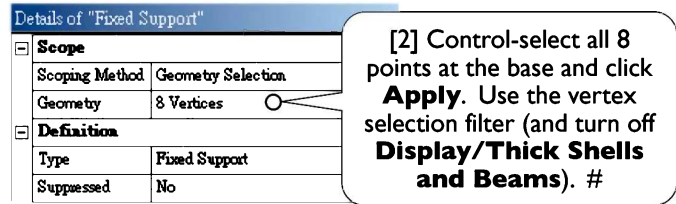
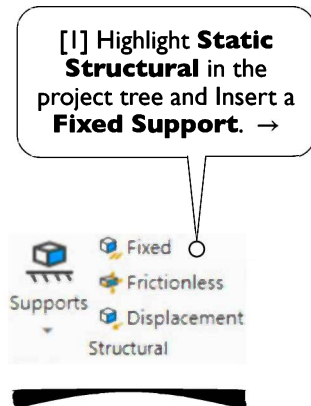


[2] The default mesh settings generate 608 elements (including shell elements and beam elements). ↗

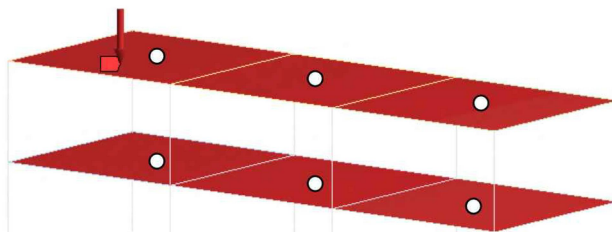
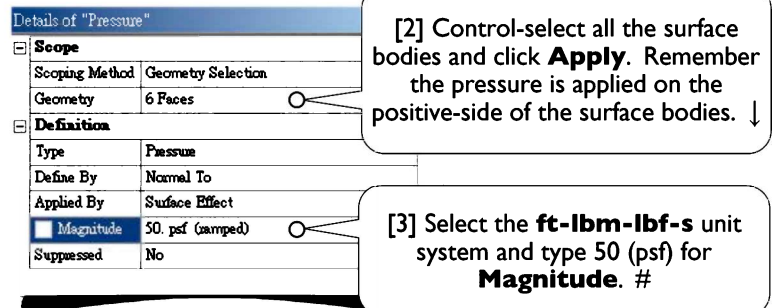
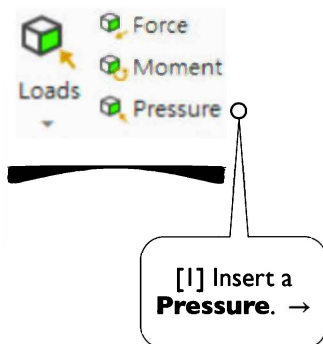


[3] Turn on **Display/ Thick Shells and Beams**. #


7.3.16 Specify Supports




7.3.17 Apply the Live Load



7.3.18 Apply Earth Gravity

 [1] Select **Inertial/Standard Earth Gravity**. ↓

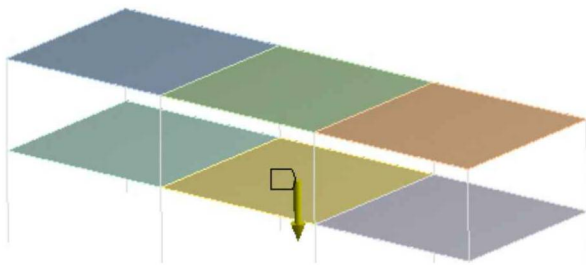
A: Static Structural
Standard Earth Gravity
Time: 1. s

 Standard Earth Gravity: 32.174 ft/s²
Components: 0, -32.174, 0 ft/s²


Details of "Standard Earth Gravity"

Scope	
Geometry	All Bodies
Definition	
Coordinate System	Global Coordinate System
X Component	0 ft/s ² (ramped)
Y Component	-32.174 ft/s ² (ramped)
Z Component	0 ft/s ² (ramped)
Suppressed	No
Direction	-Y Direction


[2] Select **-Y Direction**. This defines the direction of gravitational force. #



7.3.19 Apply the Earthquake Load

 [1] Select **Inertial/Acceleration**. ↓

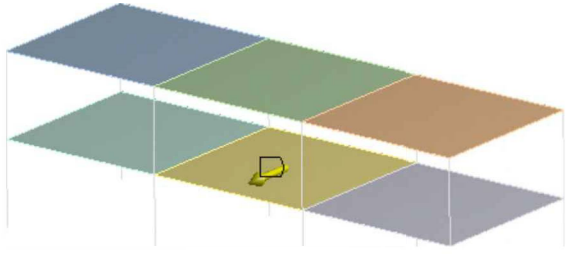
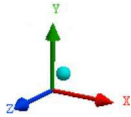
A: Static Structural
Acceleration
Time: 1. s

 Acceleration: 6.4 ft/s²
Components: 0, 0, 6.4 ft/s²

Details of "Acceleration"

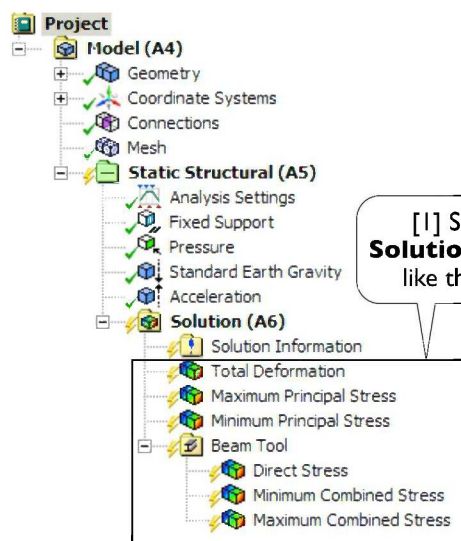
Scope	
Geometry	All Bodies
Definition	
Define By	Components <input type="radio"/>
Coordinate System	Global Coordinate System
X Component	0 ft/s ² (ramped)
Y Component	0 ft/s ² (ramped)
Z Component	6.4 ft/s ² (ramped) <input checked="" type="radio"/>
Suppressed	No

[2] Select **Components** for **Define By** and type 6.4 (ft/s²) for **Z Component**. This defines how the bodies accelerate; the "Inertia force" applies in the opposite direction (-Z direction). #

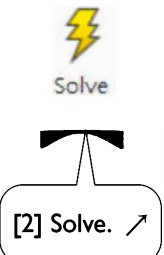



7.3.20 Solve the Model and View the Results

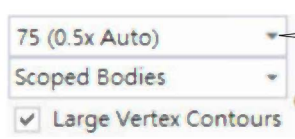
[1] Set up Solution branch like this. ✓



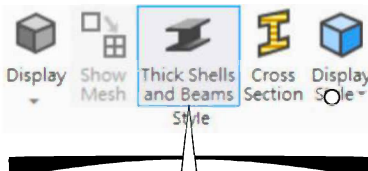
[2] Solve. ✓




[3] Select **Total Deformation. The deformation is too exaggerated; reduce the scale like this. ↓**



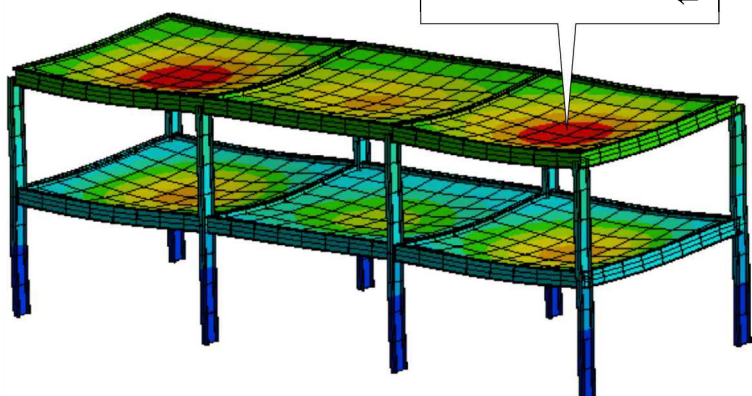
[4] Turn on **Display/Thick Shells and Beams. ↓**



[5] Select the **in-lbm-lbf-s unit system. ↓**



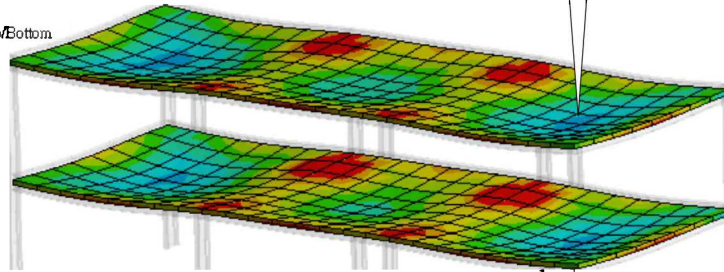
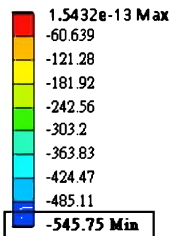
[6] The maximum deformation is 0.3 in. ↙



A: Static Structural
Total Deformation
 Type: Total Deformation
 Unit: in
 Time: 1

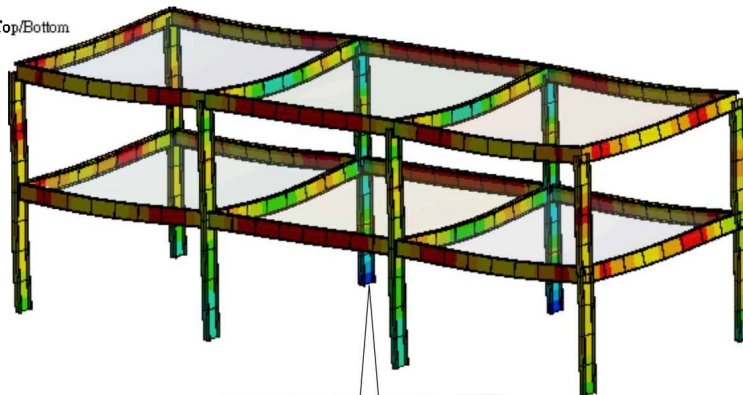
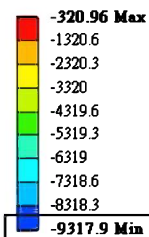
0.30373 Max
 0.26998
 0.23623
 0.20249
 0.16874
 0.13499
 0.10124
 0.067495
 0.033748
0 Min

A: Static Structural
 Minimum Principal Stress
 Type: Minimum Principal Stress - Top/Bottom
 Unit: psi
 Time: 1



[7] Maximum compressive stress of the concrete slab is about 550 psi, well below the fracture stress of the concrete (3000-4000 psi). ↓

A: Static Structural
 Minimum Combined Stress
 Type: Minimum Combined Stress - Top/Bottom
 Unit: psi
 Time: 1



[8] Maximum stress for beam/column is 9300 psi (compressive), well below the yield stress of the steel (40,000-60,000 psi). ↓

Wrap Up

[9] Save the project and exit Workbench. #

Section 7.4

Review

7.4.1 Keywords

Choose a letter for each keyword, from the list of descriptions

1. () Beam Elements
2. () Coordinates File
3. () Direct Stress and Bending Stress
4. () Maximum/Minimum Bending Stress
5. () Combined Stress
6. () Truss

Answers:

1. (A) 2. (F) 3. (C) 4. (D) 5. (E) 6. (B)

List of Descriptions

(A) A line (1D) element that can be arranged in the 3D space. It is used to mesh a body when two of its dimensions are much smaller than the third dimension. Each node has 6 degrees of freedom: 3 translational and 3 rotational. Due to the presence of rotational degrees of freedom, it is very efficient to model the problems dominated by bending modes, in contrast to a solid element, which does not have rotational degrees of freedom.

(B) Defined as a structure consisting of two-force members. By two-force member, we mean that the members are pin-jointed at the ends, and the loads apply on the joints so that the members are either stretched or compressed but not bent.

(C) The resultant forces acting on a beam cross section can be summarized into three components: direct force F , bending moment M , and shear force V . Only the direct force and the bending moment contribute to the axial stress. The axial stress caused by the direct force is called the direct stress $\sigma = F/A$, where A is the cross-sectional area. The direct stress can be tensile or compressive. The axial stress caused by the bending moment is called the bending stress $\sigma_b = My/I$, where y is the distance from the neutral axis to the point of concern, and I is the moment of inertia of the cross section.

- (D) Occurs at either the top or bottom beam edge, depending on the direction of the bending moment.
- (E) The superposition of the direct stress and the bending stress.
- (F) A text file describing the coordinates of points. The file has 5 columns, or fields: (a) group number, (b) ID number, (c) X-coordinate, (d) Y-coordinate, and (e) Z-coordinate. The group number and the ID number are arbitrary and they together uniquely identify a point from others.

7.4.2 Additional Workbench Exercises

Convergence Study of Beam Elements

Generate a convergence curve like the one in 7.1.16[2] (page 282) for the flexible gripper in Section 7.1.

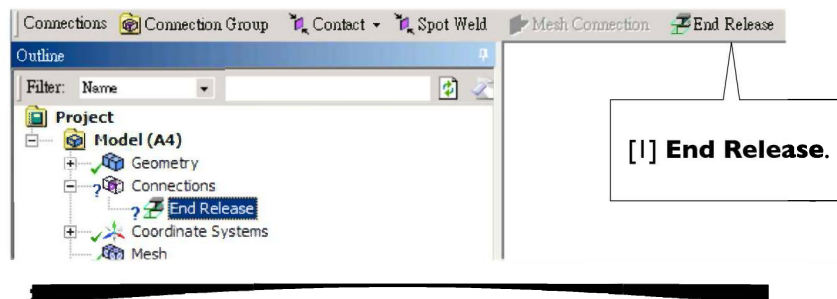
Mesh Each Member with More Elements

We mentioned in 7.2.8[4] (page 289) that the default settings of **Mesh** would mesh the model with 205 beam elements and would result in exactly the same solution as 37 elements. Verify this. Type 0 for **Element Size** to set to the default element size.

Pin-Jointed or Rigid-Jointed?

We mentioned in 7.2.1[1] (page 283) that if the members of a truss structure are slender enough, there is no essential difference between a pin-jointed model and a rigid-jointed model. Verify this. You may use the 3D truss example (Section 7.2) or select a simple truss structure from any of your Engineering Mechanics textbooks. Solve the problem with the pin-joint assumption, and then solve the problem again using a rigid-jointed model. Compare the results and draw your conclusions.

To model a pin-jointed structure in Workbench, you need to either specify revolute joints between the members or utilize **End Release**^[Ref 1] feature in **Mechanical** [1].



References

1. All Help>Mechanical Application>Mechanical User's Guide>Setting Connections>End Releases

Chapter 8

Optimization

Design Process

A typical engineering design process involves several steps as shown in [1-6]. First, the engineer sets up an initial design [1]. The design is simulated [2] and the performance is evaluated [3]. If the performance is satisfied, the design is accepted [6], otherwise the design must be improved somehow [5]. It is an iterative process. After several iterations, the design hopefully converges to an optimal design.

Conventionally, the way of improving a design [5] relies on engineers' experience. The consequence is that the process is costly and often fails to find an optimal design.

This chapter focuses on a capability of Workbench, which automates the entire design process [1-6].

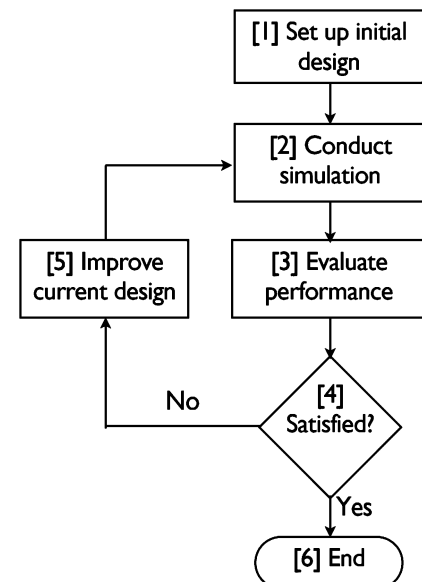
Purpose of This Chapter

This chapter is strategically arranged in the middle of the book because it is a suitable time to learn optimization techniques before we look into more advanced topics. The idea is that the students should be aware as early as possible of the optimization capabilities provided by Workbench so that they can use the features when doing their exercises.

This chapter provides two hands-on examples, which cover main ideas of optimization. After these exercises, the students should be able to use these capabilities on their own.

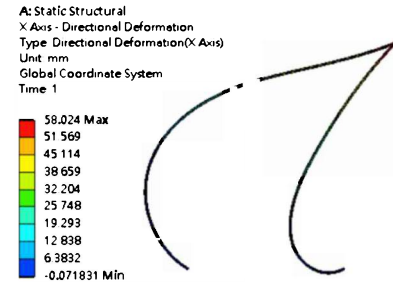
About Each Section

Section 8.1 revisits the flexible gripper, introduced in Section 7.1. This time, we want to improve the shape of the gripper to achieve an optimal geometric advantage. Section 8.2 provides an additional exercise, in which the triangular plate, introduced in Section 2.2 and simulated in Section 3.1, is to be improved to achieve a minimal weight design.



Section 8.1

Flexible Gripper^[Ref 1]



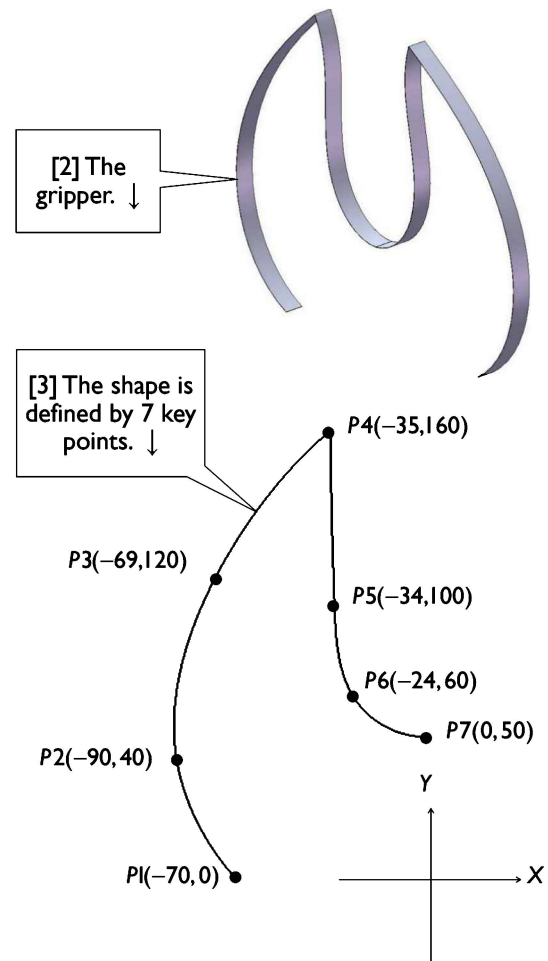
8.1.1 About the Flexible Gripper

[1] In Section 7.1, we created a model for the flexible gripper [2] and performed a static structural simulation to assess the GA (geometric advantage, defined as the ratio of the output displacement to the input displacement; see 7.1.1[1], page 272) of the design. For that particular design, the GA value is 1.04 (7.1.15[6], page 281). In this section, we want to improve the GA value by adjusting the shape of the flexible gripper.

The shape of the gripper is defined by 7 key points [3]. The positions of $P1$, $P4$, $P5$, and $P7$ are fixed (cannot be changed) due to the constraints imposed by some functional requirements. The positions of $P2$, $P3$, and $P6$ are free to be adjusted.

The idea is to fix the X-coordinates of these three points and adjust their Y-coordinates to achieve a better GA value. The allowable adjustment ranges for the Y-coordinates are ± 10 mm for $P2$, ± 20 mm for $P3$ and ± 5 mm for $P6$. Besides, there is a limitation on the stresses, either tensile or compressive, which should not exceed 15 MPa for a reliability consideration.

In Workbench, **DesignXplorer** is used to carry out the task. ↗

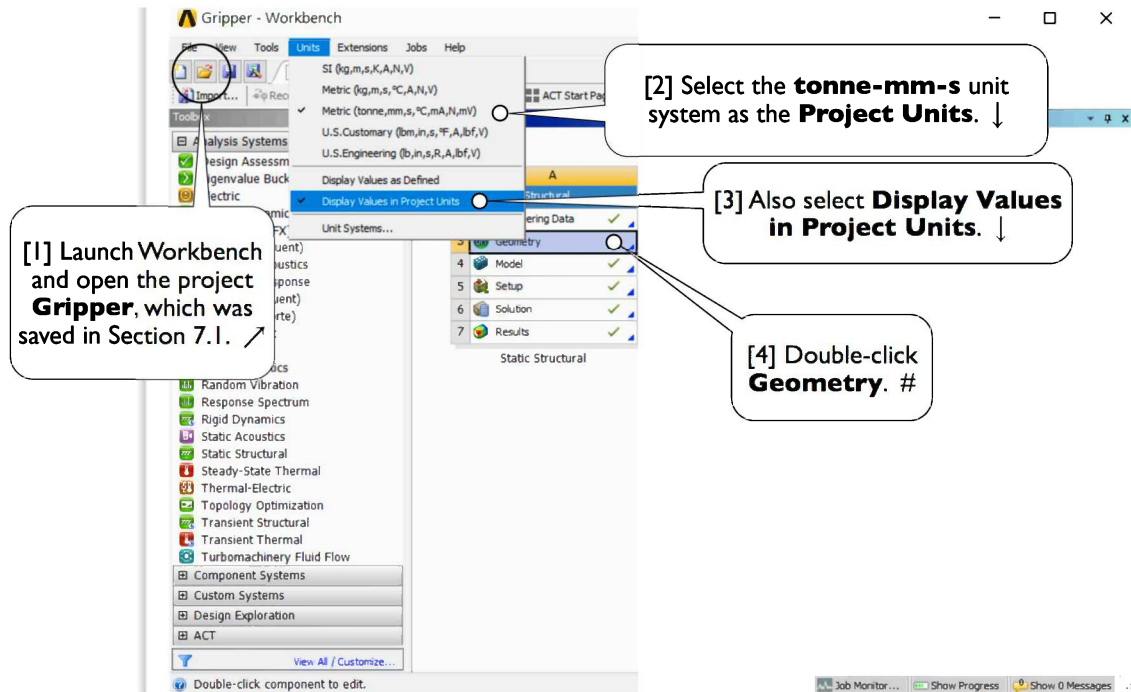


Keep the Number of Design Variables Minimum

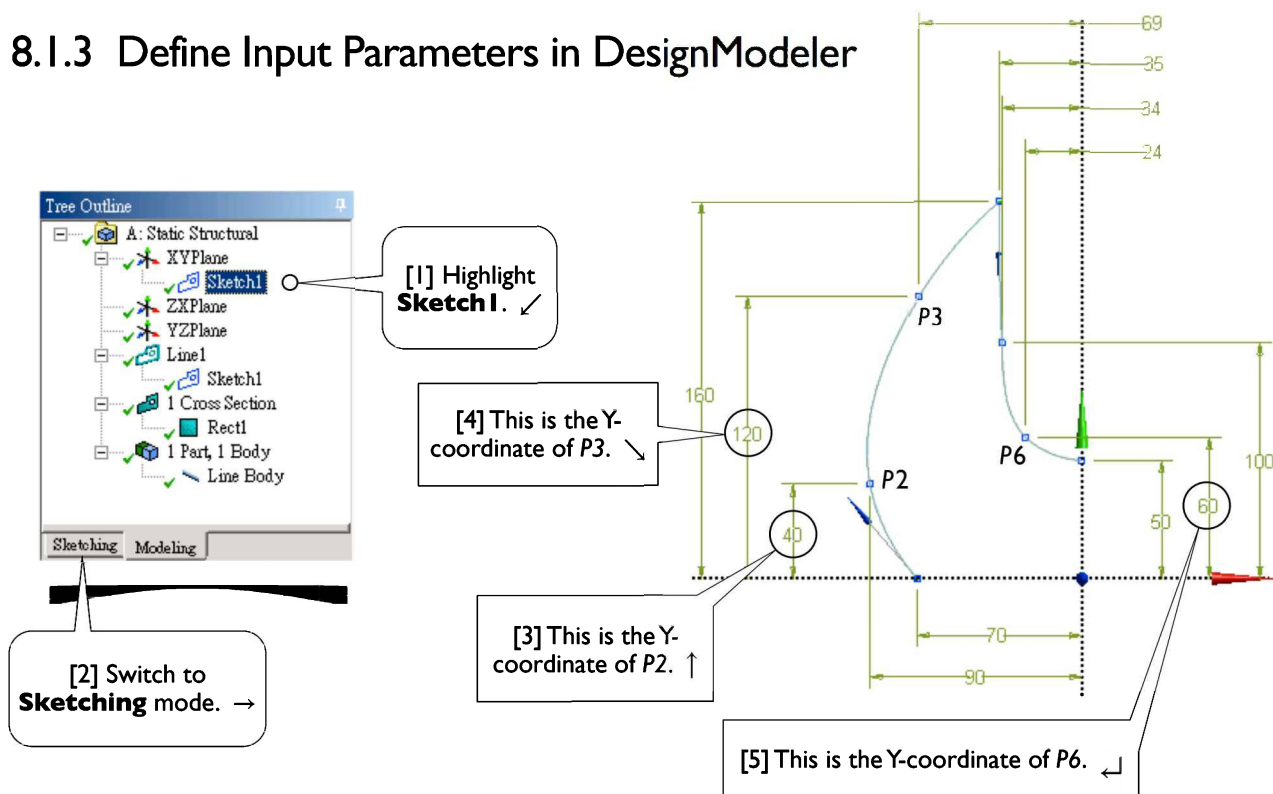
[4] The (input) parameters that can be changed to improve a design are called the *design variables* (in this case, Y-coordinates of $P2$, $P3$, and $P6$). The overall computing time and complexity of an optimization process increases dramatically as the number of design variables increases; therefore, it is important to keep the number of design variables as low as possible.

In this case, we could have chosen both X-coordinates and Y-coordinates of $P2$, $P3$, $P6$ as design variables. That would increase the number of design variables from 3 to 6 but without gaining any advantages. #

8.1.2 Resume the Project **Gripper**



8.1.3 Define Input Parameters in DesignModeler



[6] Click the box before the Y-coordinate of P2 (40 mm). Your dimension name may not be the same as here (V7). ↗

Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 12	
<input type="checkbox"/> H1	70 mm
<input type="checkbox"/> H2	90 mm
<input type="checkbox"/> H3	69 mm
<input type="checkbox"/> H4	35 mm
<input type="checkbox"/> H5	34 mm
<input type="checkbox"/> H6	24 mm
<input type="checkbox"/> V10	100 mm
<input type="checkbox"/> V11	60 mm
<input checked="" type="checkbox"/> V12	50 mm
<input checked="" type="checkbox"/> V7	40 mm
<input type="checkbox"/> V8	120 mm
<input type="checkbox"/> V9	160 mm

[7] Type **Y2** as the parameter name. ↓

A: Static Structural DesignModeler

Create a new Design parameter for dimension reference XYPlane.V7?

Parameter Name:

OK Cancel

[8] Click **OK**. ✓

[9] A **P** is marked on the box, indicating that it is an (input) parameter. Its value is controlled by the program now and cannot be manually changed unless you remove the **P** by clicking it again. →

[10] Repeat steps [6-8] for the Y-coordinates of P3 and P6 (see [11-12]). Remember that your dimension names may not be the same as they appear here. ✓

Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 12	
<input type="checkbox"/> H1	70 mm
<input type="checkbox"/> H2	90 mm
<input type="checkbox"/> H3	69 mm
<input type="checkbox"/> H4	35 mm
<input type="checkbox"/> H5	34 mm
<input type="checkbox"/> H6	24 mm
<input checked="" type="checkbox"/> V10	100 mm
<input checked="" type="checkbox"/> V11	60 mm
<input checked="" type="checkbox"/> V12	50 mm
<input checked="" type="checkbox"/> V7	40 mm
<input checked="" type="checkbox"/> V8	120 mm
<input type="checkbox"/> V9	160 mm

[12] Click the box before the Y-coordinate of P6 (60 mm) and type the name **Y6**. Close DesignModeler. ↘

- 1 Static Structural
- 2 Engineering Data ✓
- 3 Geometry ✓
- 4 Model ✓
- 5 Setup ✓
- 6 Solution ✓
- 7 Results ✓
- 8 Parameters

Static Structural

Parameter Set

[14] Double-click to open **Mechanical**. Set the unit system to **mm-kg-N-s. #**

[11] Click the box before the Y-coordinate of P3 (120 mm) and type the name **Y3** [7]. Remember your dimension name may be different from here. ↑

[13] A **Parameter Set** is added to **Project Schematic**. Parameters can be accessed from this "bus." We will explore it later (8.1.5, page 315-317) after we define additional parameters in **Mechanical**. ↑

8.1.4 Define Output Parameters in Mechanical

[1] Highlight **X Axis - Directional Deformation**.

[2] Click the box before **Maximum** under **Results**. A **P** is marked on the box, indicating that it is an (output) parameter.

[3] Highlight **Minimum Combined Stress**.

[4] Click the box before **Minimum** under **Results**.

[5] Highlight **Maximum Combined Stress**.

[6] Click the box before **Maximum** under **Results**. Close **Mechanical**. #

Results	
Minimum	-1.9668e-003 mm
P Maximum	52.035 mm
Average	19.582 mm
Minimum Occurs On	Line Body
Maximum Occurs On	Line Body

Results	
P Minimum	-12.508 MPa
Maximum	-2.9954e-002 MPa
Average	-5.1514 MPa
Minimum Occurs On	Line Body
Maximum Occurs On	Line Body

Results	
Minimum	-8.8344e-003 MPa
P Maximum	12.478 MPa
Average	5.1437 MPa
Minimum Occurs On	Line Body
Maximum Occurs On	Line Body

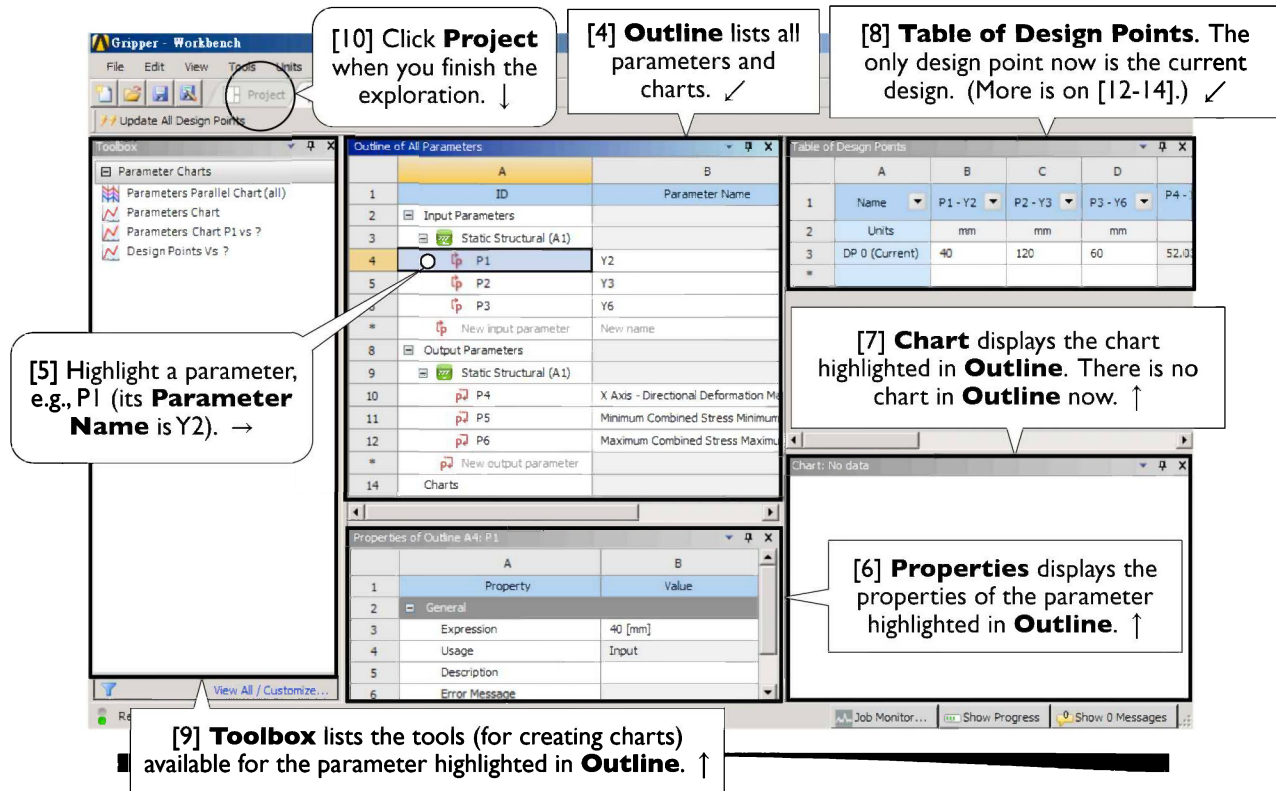
8.1.5 Explore the Parameter Set

[1] Double-click **Parameter Set**.

[2] Select **View/Reset Workspace** to make sure your workspace is the "standard" workspace (i.e., the GUI).

[3] Note that there are several window panes in the workspace. Each can be turned on/off.

View	Tools	Units	Extension
Refresh			F5
Reset Workspace			
Reset Window Layout			
✓ Toolbox			
Toolbox Customization			
✓ Outline			
✓ Properties			
✓ Table			
✓ Chart			
Messages			
Progress			
Sidebar Help			



Notations Used in This Section

[11] Workbench uses P1, P2, P3 (non-italic) and so on as parameter names [5]. In our case of the flexible gripper, we use P2, P3, and P6 (italic) for three control points of the shape. Be careful not to confuse these two sets of notations.

Input Parameters and Output Parameters

In optimization jargon, input parameters are also called *design variables*. Initial values of input parameters are set up by the user, and Workbench automatically changes these values to improve the design according to the algorithm used. The values of output parameters are calculated from the simulation applications (e.g., **Mechanical**). In our case, the input parameters are Y2 (P1), Y3 (P2), and Y6 (P3), and the output parameters are the maximum horizontal displacement (P4), the minimum combined stress (P5), and the maximum combined stress (P6). Note that the input parameters are not necessarily defined in DesignModeler. For example, the loads defined in **Mechanical** could be defined as input parameters.

Design Space

The space spanned by the input parameters is called the *design space*. In our case, the space spanned by parameters Y2, Y3, and Y6 is the design space, which is a three-dimensional space. The optimization process can be thought of as a process of searching for an optimal point in the design space.

Design Points

Any point in the design space is called a *design point*. **Table of Design Points** [8] lists a series of calculated design points. Initially, we have only one design point: Y2 = 40, Y3 = 120, Y6 = 60. This design point is now called the *current design* [8]. **Current Design** is a design point of which the data are stored in **Mechanical** database. ↩

Table of Design Points							
	A	B	C	D	E	F	G
1	Name ▾	P1 - Y2 ▾	P2 - Y3 ▾	P3 - Y6 ▾	P4 - X Axis - Directional Deformation Maximum ▾	P5 - Minimum Combined Stress Minimum ▾	P6 - Maximum Combined Stress Maximum ▾
2	Units	mm	mm	mm	mm	MPa	
3	Current	40	120	60	52.035	-12.508	12.478
4	DP 1	45	110	57	⚡	⚡	⚡
5	DP 2	47	125	62	⚡	⚡	⚡
6	DP 3	35	105	60	⚡	⚡	⚡
*							

[12] A useful feature of **Table of Design Points** [8] is that you can type values for input parameters... ↓

[13] And click **Update All Design Points** to evaluate the output parameters. (A demonstration is given in 10.1.8, page 372.) →

⚡ Update All Design Points

[14] You can also select one or more design points and right-click-select **Update Selected Design Points**. #

8.1.6 Create a Direct Optimization System

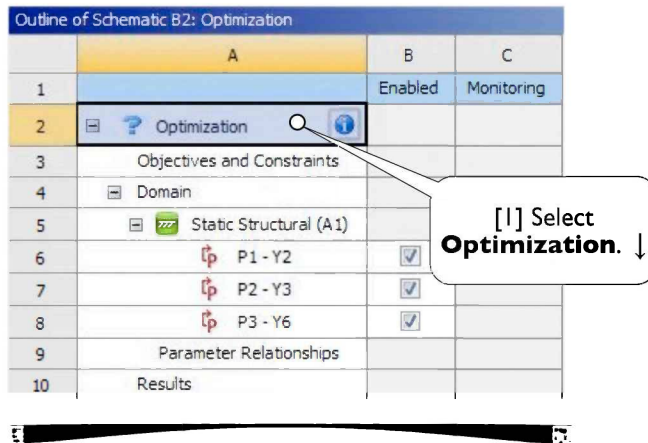
The screenshot shows the Gripper Workbench interface. On the left is the 'Toolbox' with various analysis systems. In the center is the 'Project Schematic' showing a hierarchy of components: A (Static Structural), B (Parameter Set), and C (Direct Optimization). A red arrow points from the 'Parameters' component in A to the 'Direct Optimization' component in C. On the right is the 'Design Exploration' tool, which is currently expanded.

[1] Expand **Design Exploration** and double-click **Direct Optimization**. →

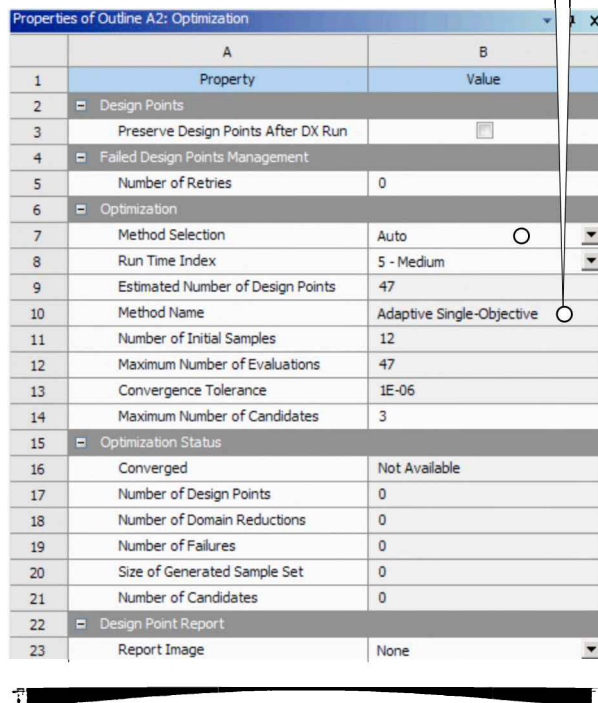
[2] A **Direct Optimization** system is created, and a link indicates that information can be passed between them. ↓

[3] Double-click **Optimization**. #

8.1.7 Set Up Optimization Method^[Ref 2]



[2] By default, the **Adaptive Single-Objective** method is used. The optimization methods available are listed in [3]. →



Optimization Methods

NLPQL

[3] The method (Nonlinear Programming by Quadratic Lagrangian) is a gradient-base algorithm to provide a local optimization result. It supports single objective and is limited to continuous input parameters.

Here, we select this method simply because it is one of the classical optimization methods. After completing this exercise, you are encouraged to try other methods.

Adaptive Single-Objective

It is also a gradient-based algorithm, providing a global optimization result. It is limited to single objective; it supports continuous and manufacturable input parameters.

Adaptive Multiple-Objective

It supports multiple objectives and aims at finding the global optimum. It supports continuous and manufacturable input parameters.

MISQP

The method (Mixed-Integer Sequential Quadratic Programming) solves mixed-integer nonlinear programming problems by a modified SQP method.

Screening

It uses a simple approach based on sampling and sorting. It supports multiple objectives as well as all types of input parameters. It is usually used for preliminary design, e.g., global behavior study.

MOGA

The method (Multi-Objective Genetic Algorithm) supports multiple objectives and aims at finding global optimum. #

8.1.8 Set Up Objective and Constraints

Outline of Schematic B2: Optimization

	A	B	C
1		Enabled	Monitoring
2	Optimization		
3	Objectives and Constraints		
4	Domain		
5	Static Structural (A1)		
6	P1 - Y2	<input checked="" type="checkbox"/>	
7	P2 - Y3	<input checked="" type="checkbox"/>	
8	P3 - Y6	<input checked="" type="checkbox"/>	
9	Parameter Relationships		
10	Results		

[1] Select **Objectives and Constraints**.

[2] In the **Table of Schematic**, set up **Parameter**, **Objective**, and **Constraint** like this. Note that the unit for stress is **MPa**. #

Table of Schematic B2: Optimization

	A	B	C	D	E	F	G	H	I
1	Name	Parameter	Objective	Target	Tolerance	Type	Lower Bound	Upper Bound	Tolerance
2			Type						
3	Maximize P4	P4 - X Axis - Directional Deformation Maximum	Maximize	0		No Constraint			
4	P5 >= -15 MPa	P5 - Minimum Combined Stress Minimum	No Objective			Values >= Lower Bound	-15		0.001
5	P6 <= 15 MPa	P6 - Maximum Combined Stress Maximum	No Objective			Values <= Upper Bound		15	0.001
*		Select a Parameter							

8.1.9 Set Up Lower/Upper Bounds for Input Parameters

Properties of Outline A9: P1 - Y2

	A	B
1	Property	Value
2	General	
3	Units	mm
4	Classification	Continuous
5	Values	
6	Lower Bound	30
7	Upper Bound	50
8	Allowed Values	Any

[2] Type 30 for **Lower Bound** and 50 for **Upper Bound**. →

Properties of Outline A10: P2 - Y3

	A	B
1	Property	Value
2	General	
3	Units	mm
4	Classification	Continuous
5	Values	
6	Lower Bound	100
7	Upper Bound	140
8	Allowed Values	Any

[4] Type 100 for **Lower Bound** and 140 for **Upper Bound**. →

Outline of Schematic B2: Optimization

	A	B	C
1		Enabled	Monitoring
2	Optimization		
3	Objectives and Constraints		
4	Maximize P4		
5	P5 >= -15 MPa		
6	P6 <= 15 MPa		
7	Domain		
8	Static Structural (A1)		
9	P1 - Y2	<input checked="" type="checkbox"/>	
10	P2 - Y3	<input checked="" type="checkbox"/>	
11	P3 - Y6	<input checked="" type="checkbox"/>	
12	Parameter Relationships		
13	Results		

[1] In **Outline**, select **P1-Y2**. ←

[3] Select **P2-Y3**. ✓

[5] Select **P3-Y6**. ↓

Properties of Outline A11: P3 - Y6

	A	B
1	Property	Value
2	General	
3	Units	mm
4	Classification	Continuous
5	Values	
6	Lower Bound	55
7	Upper Bound	65
8	Allowed Values	Any

[6] Type 55 for **Lower Bound** and 65 for **Upper Bound**. #

8.1.10 Run the Optimization

[1] Click **Update**. See [2-5] while the program is running. →

[2] **Table of Schematic** displays the values of input/output parameters in each evaluation of the output parameters. ↓

[3] While the optimization is running, click **Show Progress**. ←

[4] A **Progress** window reports the status. It takes a while to complete the optimization process. ↘

	A	B	C	D	E	F	G
	Name	P1 - Y2 (mm)	P2 - Y3 (mm)	P3 - Y6 (mm)	P4 - Y Axis - Directional Deformation Maximum (mm)	P5 - Minimum Combined Stress Minimum (MPa)	P6 - Maximum Combined Stress Maximum (MPa)
1							
2	1 DP	39.167	121.67	60.417	51.807	-12.494	12.467
3	2 DP	47.5	131.67	57.917	50.954	-11.495	11.458
4	3 DP	30.833	111.67	61.25	54.037	-13.504	13.477
5	4 DP	34.167	138.33	59.583	54.929	-11.747	11.719
6	5 DP	35.833	125	64.583	50.877	-12.76	12.761
7	6 DP	32.5	118.33	56.25	53.995	-12.186	12.133
8	7 DP	42.5	105	63.75			
9	8 DP	49.167	115	62.083			
10	9 DP	37.5	101.67	58.75			
11	10 DP 1	44.167	135	62.917			
12	11 DP 1	40.833	128.33	55.417			

8.1.11 Retrieve the Optimal Design

	A	B	C
1	Enabled	Monitoring	
2	✓ Optimization		
3	Objectives and Constraints		
4	Maximize P4		
5	P5 >= -15 MPa		
6	P6 <= 15 MPa		
7	Domain		
8	Static Structural (A1)		
9	P1 - Y2	✓	
10	P2 - Y3	✓	
11	P3 - Y6	✓	
12	Parameter Relationships		
13	Raw Optimization Data		
14	Convergence Criteria		
15	Results		
16	✓ Candidate Points		
17	✓ Tradeoff		
18	✓ Samples		

[1] In **Outline**, click **Candidate Points**. ←

[5] When the optimization is completed, click **Hide Progress**. #

[3] The values of the input parameters. ↓

	A	B	P1 - Y2 (mm)	P2 - Y3 (mm)	P3 - Y6 (mm)	P4 - X Axis - Directional Deformation Maximum (mm)	P5 - Minimum Combined Stress Minimum (MPa)	P6 - Maximum Combined Stress Maximum (MPa)
1	Reference	Name				Parameter Value	Variation from Reference	Parameter Value
2								
3	●	Candidate Point 1	30	100	55	58.024	0.00%	13.144
4	○	Candidate Point 2	32.122	102.76	57.574	56.507	-2.61%	13.484
5	○	Candidate Point 3	37.5	101.67	58.75	55.829	-3.38%	13.643
*		New Custom Candidate Point	40	120	60			

[4] Now, we want to copy this design to the **Table of Design Points** (8.1.5[8], page 316). Right-click here and select **Insert as Design Point**. →

[2] **Candidate Point 1** gives the best GA value ($58.024/50 = 1.16$). For a definition of GA value, see 7.1.1[1], page 272. ↖

[5] Click **Project** to return to **Project Schematic**. #

8.1.12 Set the Optimal Design as Current Design

[1] Double-click **Parameter Set**. ↘

[5] Right-click here and select **Update Selected Design Points**. ↘

[4] The **Candidate Point 1** becomes the current design. ←

	A	B	C	D	E	F	G
1	Name	P1 - Y2	P2 - Y3	P3 - Y6	P4 - X Axis - Directional Deformation Maximum	P5 - Minimum Combined Stress Minimum	P6 - Maximum Combined Stress Maximum
2	Units	mm	mm	mm	mm	MPa	MPa
3	DP 0 (Current)	30	100	55	58.024	-13.211	13.144
4	DP 1	30	100	55			
*							

[3] Right-click here and select **Copy inputs to Current**. ↗

[2] The **Candidate Point 1** (see 8.1.1[4], this page) was inserted here. ←

[6] The output parameters are updated. ↓

[7] Click **Project** to return to **Project Schematic**. #

8.1.13 View the Current Design

[1] Double-click **Geometry**. ↓

[4] Double-click **Model**. ↘

[3] The new design. Close DesignModeler. ↑

[2] Switch to the **Sketching** mode and notice that these dimensions have been updated. ↑

[5] The simulation results are also updated. Close **Mechanical**. ↙

Details View

Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 12	
<input type="checkbox"/> H1	70 mm
<input type="checkbox"/> H2	90 mm
<input type="checkbox"/> H3	69 mm
<input type="checkbox"/> H4	35 mm
<input type="checkbox"/> H5	34 mm
<input type="checkbox"/> H6	24 mm
<input type="checkbox"/> V10	100 mm
<input checked="" type="checkbox"/> V11	55 mm
<input type="checkbox"/> V12	50 mm
<input checked="" type="checkbox"/> V7	30 mm
<input checked="" type="checkbox"/> V8	100 mm
<input type="checkbox"/> V9	160 mm

A: Static Structural
X Axis - Directional Deformation
 Type Directional Deformation(X Axis)
 Unit mm
 Global Coordinate System
 Time 1

58.024 Max
 51.569
 45.114
 38.659
 32.204
 25.748
 19.293
 12.838
 6.3832
 -0.071831 Min

Wrap Up

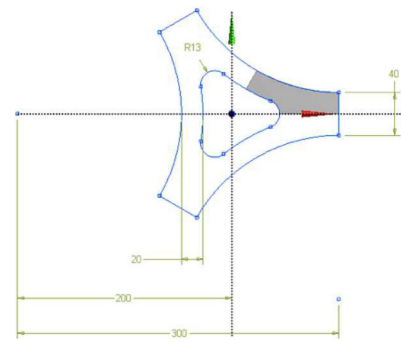
[6] Save the project and exit Workbench. #

Reference

1. Chao-Chieh Lan and Yung-Jen Cheng, 2008, "Distributed Shape Optimization of Compliant Mechanisms Using Intrinsic Functions," *ASME Journal of Mechanical Design*, Vol. 130, 072304.
2. All Help>DesignXplorer>ANSYS DesignXplorer User's Guide>Using Goal Driven Optimization

Section 8.2

Triangular Plate



8.2.1 About the Triangular Plate

[1] In Section 3.1, we generated a 2D solid model and performed a 2D static simulation for the triangular plate [2-5] introduced in Section 2.2.

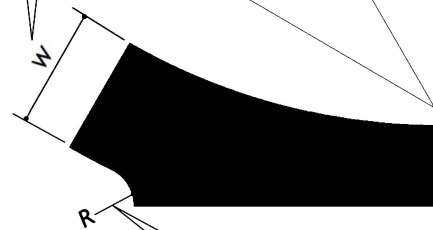
The plate is made of steel with an allowable stress of 100 MPa. According to the simulation results in Section 3.1, the initial design gives a maximum stress of about 51 MPa (3.1.13[2], page 122), well below the allowable stress. That means the initial design is over-designed: the material can be cut down somehow. In this section, we want to redesign the triangular plate to reduce the amount of material.

The design variables (input parameters) are the width of the bridges [4] and the radius of the fillets [5]. For the width W , the initial design is 30 mm and its allowable range is 20-30 mm. For the radius R , the initial design is 10 mm and its allowable range is 5-15 mm. The project unit system used in this exercise is **tonne-mm-s**. ↗

[2] The triangular plate. →

[3] One-sixth of the model will be analyzed due to the symmetries. ✓

[4] The initial design of the width of the bridges is 30 mm. ↓

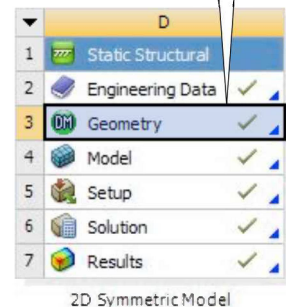
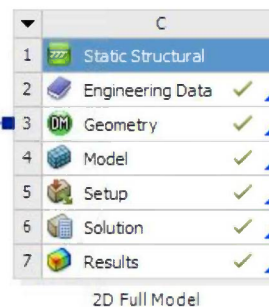
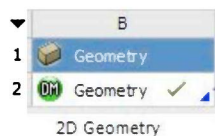
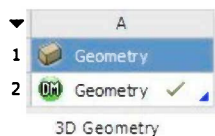


[5] The initial design of the radius of the fillets is 10 mm. #

8.2.2 Resume Project **Triplate**

[1] Launch Workbench. Open the project **Triplate**, which was saved in Section 3.1. Select **tonne-mm-s** as the project units and make sure **Display values in Project Units** is selected (see 8.1.2[2-3], page 313). →

[2] In the **2D Symmetric Model** system, double-click **Geometry**. #

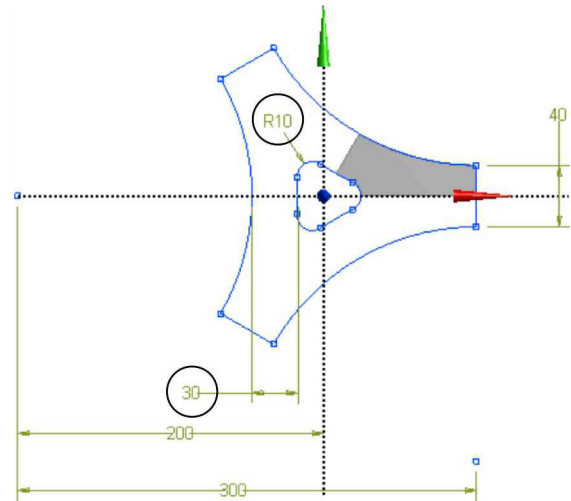


8.2.3 Define Input Parameters in DesignModeler

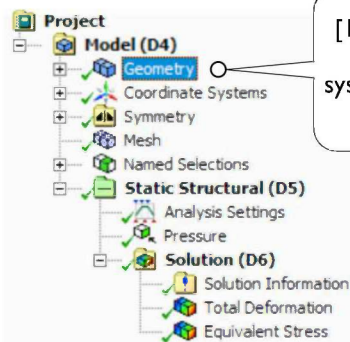
[1] Select **Sketch1**. In **Details View** of **Sketch1**, select the width (30 mm) as an input parameter and type the name **Width**. Your dimension names may not be the same as here. ↓

[2] Select the radius of the fillet (10 mm) as another input parameter and type the name **Radius**. Close DesignModeler. #

Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 5	
<input type="checkbox"/> H1	300 mm
<input type="checkbox"/> H2	200 mm
<input checked="" type="checkbox"/> H4	30 mm
<input checked="" type="checkbox"/> R5	10 mm
<input type="checkbox"/> V3	40 mm



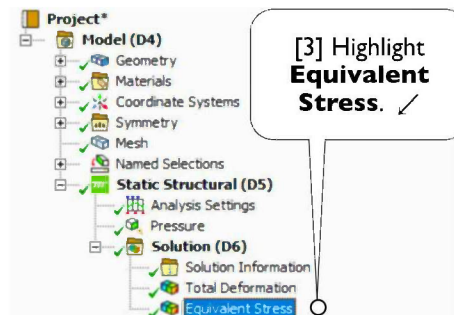
8.2.4 Define Output Parameters in Mechanical



[1] Double-click **Model** of the **2D Symmetric Model** system to start up **Mechanical**. Highlight **Geometry**. ✓

Details of "Geometry"	
Definition	
Source	C:\Users\ASUS\Docu...
Type	DesignModeler
Length Unit	Meters
Element Control	Program Controlled
2D Behavior	Plane Stress
Display Style	Body Color
Bounding Box	
Properties	
<input type="checkbox"/> Volume	22967 mm³
<input checked="" type="checkbox"/> Mass	0.18029 kg
Surface Area(approx.)	2296.7 mm²
Scale Factor Value	1.

[2] Select **Mass** as an output parameter. ✓

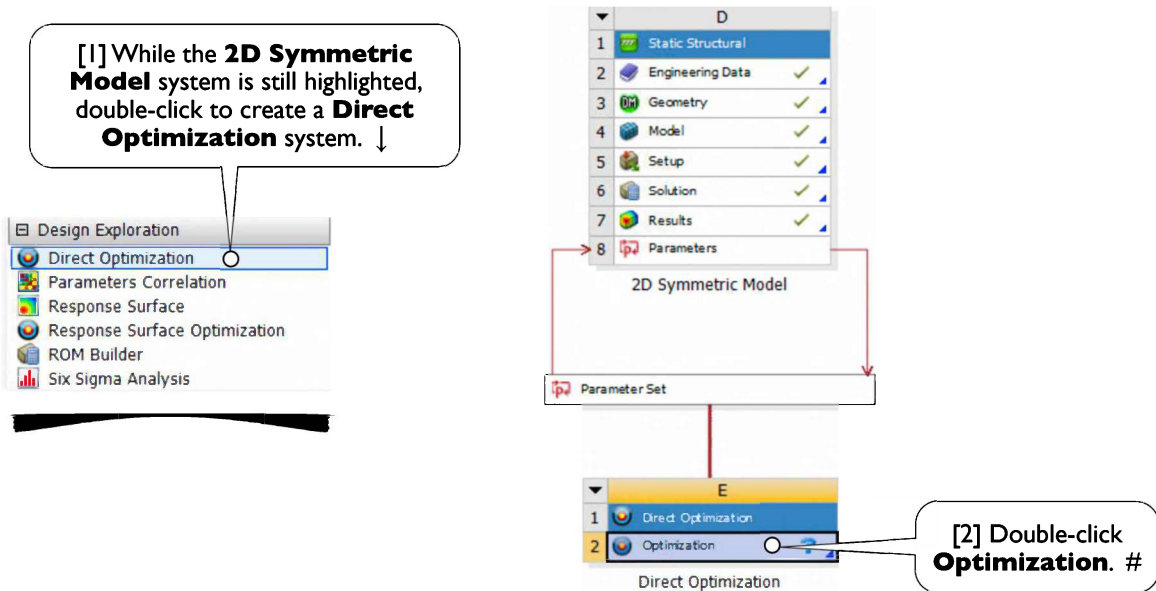


[3] Highlight **Equivalent Stress**. ✓

Details of "Equivalent Stress"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Equivalent (von-Mises) Stress
By	Time
<input type="checkbox"/> Display Time	Last
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Results	
<input type="checkbox"/> Minimum	18.934 MPa
<input checked="" type="checkbox"/> Maximum	57.164 MPa
<input type="checkbox"/> Average	38.605 MPa
Minimum Occurs On	Surface Body
Maximum Occurs On	Surface Body
Information	

[4] Select **Maximum** as an output parameter. Close **Mechanical**. #

8.2.5 Create a Direct Optimization System



8.2.6 Define and Run the Optimization Problem

[1] In **Outline**, click **Optimization**. ↓

[2] By default, **Adaptive Single-Objective** is used. This method supports manufacturable input parameters (8.1.7[3], page 318). ←

[3] Click **Objectives and Constraints**. ↓

[4] Set up objectives and constraints like this. ↙

Table of Schematic E2: Optimization									
	A	B	C	D	E	F	G	H	I
1	Name	Parameter	Objective		Constraint				
2			Type	Target	Tolerance	Type	Lower Bound	Upper Bound	Tolerance
3	Minimize P3	P3 - Geometry Mass	Minimize	0		No Constraint			
4	P4 ≤ 100 MPa	P4 - Equivalent Stress Maximum	No Objective			Values ≤ Upper Bound		100	0.001
*		Select a Parameter							

Outline of Schematic E2: Optimization			
	A	B	C
1		Enabled	Monitoring
2	Optimization		
3	Objectives and Constraints		
4	Minimize P3		
5	P4 ≤ 100 MPa		
6	Domain		
7	2D Symmetric Model (D1)		
8	P1 - Width	<input type="radio"/>	<input checked="" type="checkbox"/>
9	P2 - Radius	<input type="radio"/>	<input checked="" type="checkbox"/>
10	Parameter Relationships		
11	Results		

[5] In **Outline**, click **P1 - Width**. ↓

[8] Click **P2 - Radius**. ↓

Properties of Outline A8: P1 - Width			
	A	B	
1	Property	Value	
2	General		
3	Units	mm	
4	Classification	Continuous	
5	Values		
6	Lower Bound	20	<input type="radio"/>
7	Upper Bound	30	<input type="radio"/>
8	Allowed Values	Manufacturable Values	
9	Number Of Levels	2	

[6] In **Properties**, type 20 (mm) for **Lower Bound** and 30 (mm) for **Upper Bound**. Select **Manufacturable Values** for **Allowed Values**. ←

Table of Outline A8: P1 - Width		
	A	B
1	Name	Manufacturable Values (mm)
2	Level 1	20
3	Level 2	21
4	Level 3	22
5	Level 4	23
6	Level 5	24
7	Level 6	25
8	Level 7	26
9	Level 8	27
10	Level 9	28
11	Level 10	29
12	Level 11	30
*	New Level	

[7] In **Table of Outline**, enter 21, 22, 23, 24, 25, 26, 27, 28, and 29 for **Manufacturable Values**. A total of 11 levels are specified. Each value must be entered in the box next to **New Level**. ↑

Properties of Outline A9: P2 - Radius			
	A	B	
1	Property	Value	
2	General		
3	Units	mm	
4	Classification	Continuous	
5	Values		
6	Lower Bound	5	<input type="radio"/>
7	Upper Bound	15	<input type="radio"/>
8	Allowed Values	Manufacturable Values	
9	Number Of Levels	2	

[7] type 5 (mm) as **Lower bound** and 15 (mm) as **Upper Bound**. Select **Manufacturable Values** for **Allowed Values**. ←

Table of Outline A9: P2 - Radius		
	A	B
1	Name	Manufacturable Values (mm)
2	Level 1	5
3	Level 2	6
4	Level 3	7
5	Level 4	8
6	Level 5	9
7	Level 6	10
8	Level 7	11
9	Level 8	12
10	Level 9	13
11	Level 10	14
12	Level 11	15
*	New Level	

[10] In **Table of Outline**, enter 6, 7, 8, 9, 10, 11, 12, 13, and 14 for **Manufacturable Values**. A total of 11 levels are specified. Remember that each value must be entered in the box next to **New Level**. ↘

Update

[11] Click **Update**. It takes a while to complete the optimization. #

8.2.7 View the Optimal Design

Outline of Schematic E2: Optimization

	A	B	C
1		Enabled	Monitoring
2	✓ Optimization		
3	Objectives and Constraints		
4	Minimize P3		
5	P4 ≤ 100 MPa		
6	Domain		
7	2D Symmetric Model (D1)		
8	P1 - Width	✓	
9	P2 - Radius	✓	
10	Parameter Relationships		
11	✓ Raw Optimization Data		
12	✓ Convergence Criteria		
13	Results		
14	✓ Candidate Points		
15	✓ Tradeoff		
16	✓ Samples		

[1] In **Outline**, click **Candidate Points**. →

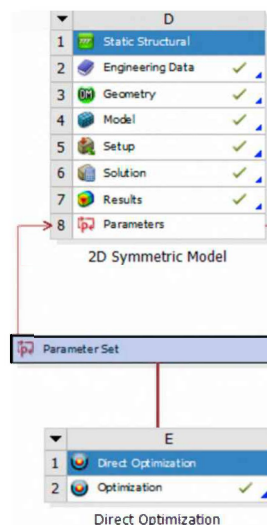
[2] **Candidate Point 1** has the minimal weight. ✓

Table of Schematic E2: Optimization , Candidate Points

	A	B	C	D	E	F	G	H
1	Reference	Name	P1 - Width (mm)	P2 - Radius (mm)	P3 - Geometry Mass (tonne)	P4 - Equivalent Stress Maximum (MPa)		
2					Parameter Value	Variation from Reference	Parameter Value	Variation from Reference
3	○	Candidate Point 1	20	13	✖✖ 0.00014833	0.00%	★ ★ ★ 97.118	0.00%
4	○	Candidate Point 2	21	14	✖✖ 0.00015408	3.88%	★ ★ ★ 85.398	-12.07%
5	○	Candidate Point 3	22	11	✖✖ 0.00015462	4.24%	★ ★ ★ 89.781	-7.55%
*		New Custom Candidate Point	20	5				

[3] Right-click here and select **Insert as Design Point**. ↓

[4] Click **Project** to return to **Project Schematic**. →



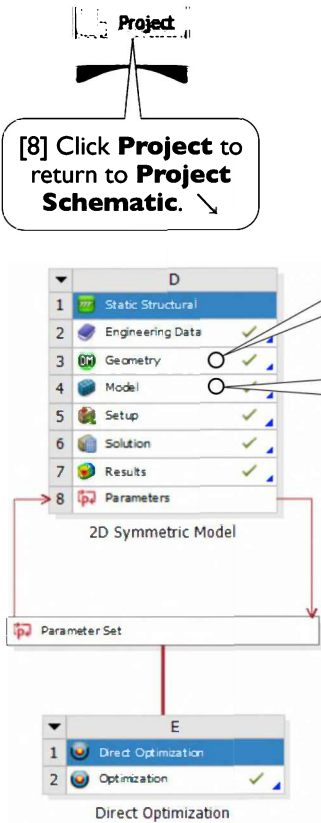
[5] Double-click **Parameter Set**. ↩

[7] Right-click here and select **Update Selected Design Points.** ↓

[6] Right-click here and select **Copy inputs to Current.** ↑

Table of Design Points					
	A	B	C	D	E
1	Name	P1 - Width	P2 - Radius	P3 - Geometry Mass	P4 - Equivalent Stress Maximum
2	Units	mm	mm	tonne	MPa
3	DP 0 (Current)	20	13	0.00014833	97.118
4	DP 1	20	13	⚡	⚡
*					

[8] Click **Project** to return to **Project Schematic.** ↘



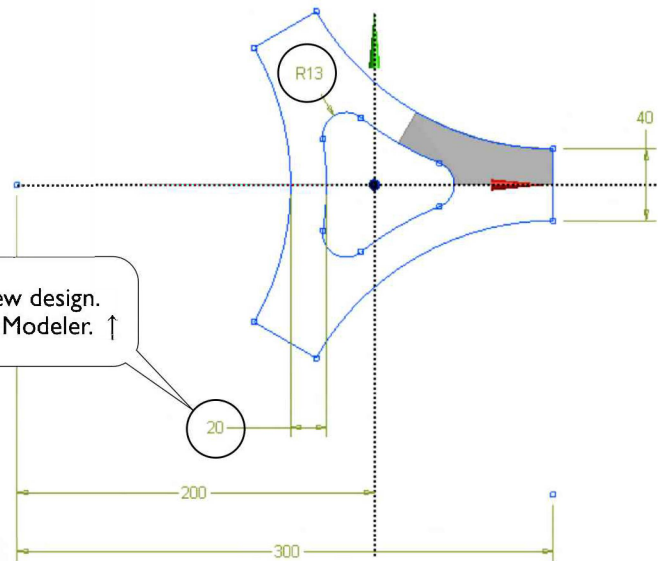
[9] Double-click **Geometry.** →

[12] Double-click **Model.** ↓

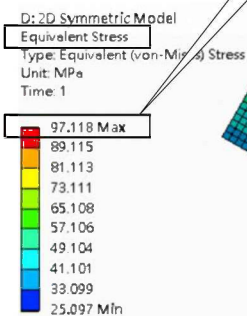
Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 5	
H1	300 mm
H2	200 mm
D H4	20 mm
D R5	13 mm
V3	40 mm

[10] Select **Sketch1.** Switch to the **Sketching** mode and notice that these dimensions have been updated. ✓

[11] The new design. Close DesignModeler. ↑



[13] The simulation results are also updated. Close **Mechanical.** ↘



Wrap Up

[14] Save the project and exit Workbench. #

Section 8.3

Review

8.3.1 Keywords

Choose a letter for each keyword, from the list of descriptions

1. () Design Points
2. () Design Space
3. () Input Parameters and Output Parameters
4. () NLPQL

Answers:

1. (C) 2. (B) 3. (A) 4. (D)

List of Descriptions

(A) Initial values of input parameters, also called design variables, are set up by the user and subsequently updated by Workbench. Values of output parameters, also called state variables, are calculated from the simulation applications.

(B) The space spanned by the input parameters, or design variables.

(C) Any points in the design space. In **DesignXplorer**, they are those with which the simulations have been carried out.

(D) Short for nonlinear programming by quadratic Lagrangian. A method of finding an optimal design. In each iteration, it approaches the problem using a quadratic polynomial and finds the optimal design in the subproblem described by the quadratic polynomial. The process is repeated until the optimal design is found. The method deals with only single-objective problems.

Chapter 9

Meshing

So far, we haven't discussed much on meshing because, for linear static problems, we usually can obtain solutions which are acceptable both in computing time and accuracy with global mesh controls (such as Relevance Center, Relevance, and **Element Size**) and some simple local mesh controls (such as **Sizing**). For the rest of the book, we will be dealing with dynamic and nonlinear problems, of which the solutions are sensitive to meshing quality. With poor mesh quality, a simulation may end up with solutions of poor accuracy or even run into convergence problems. Dynamic and nonlinear problems require many computational resources, and poor mesh quality may aggravate the situation and result in a lengthy computing time. For nonlinear problems, it is possible to reduce the runtime by improving the mesh quality (because they converge easier). In contrast, it is possible that a nonlinear solution fails to converge just because of the poor mesh quality.

Purpose of This Chapter

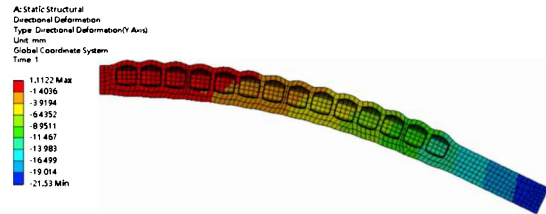
This chapter introduces meshing methods provided by Workbench and demonstrates how to use them. One of the mesh quality metrics, called *skewness*, is also introduced and used as a measure of mesh quality in this chapter.

About Each Section

Using the pneumatic finger example, Section 9.1 provides a step-by-step exercise to introduce some important concepts of meshing technologies. Section 9.2 uses a more involved model, the cover of pressure cylinder, to provide more exercises on meshing technologies. Section 9.3, a sequel of Section 3.5, studies 3D elements convergence behaviors. We postponed the study of 3D elements convergence until now because we need more meshing techniques to control the meshing density.

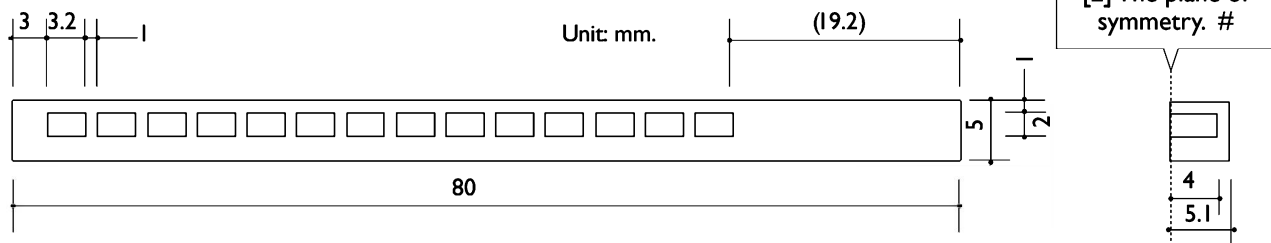
Section 9.1

Pneumatic Fingers^[Ref 1]

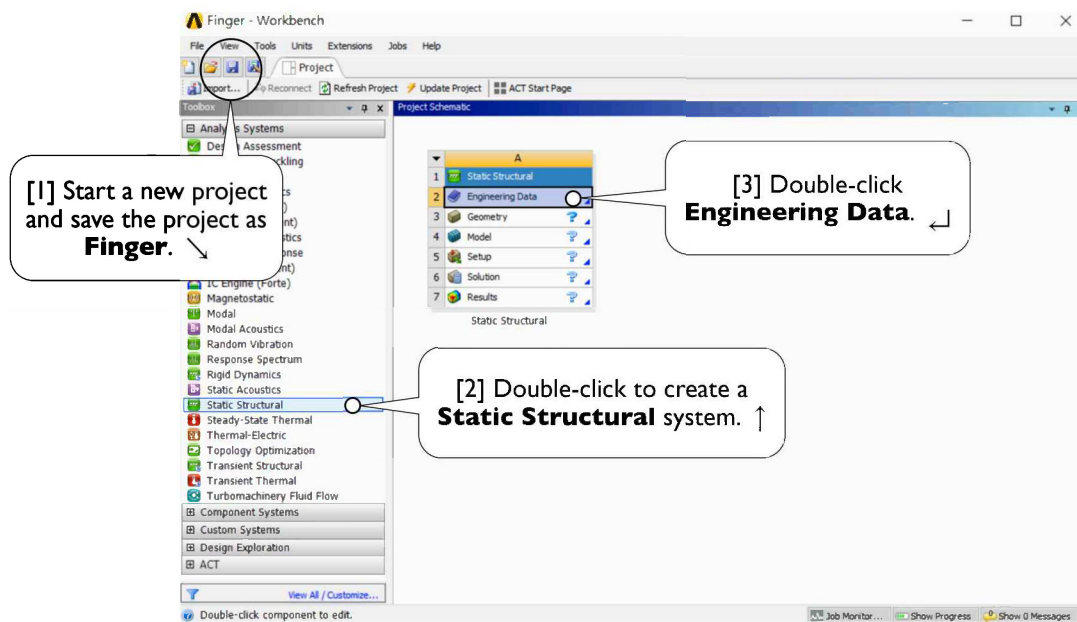


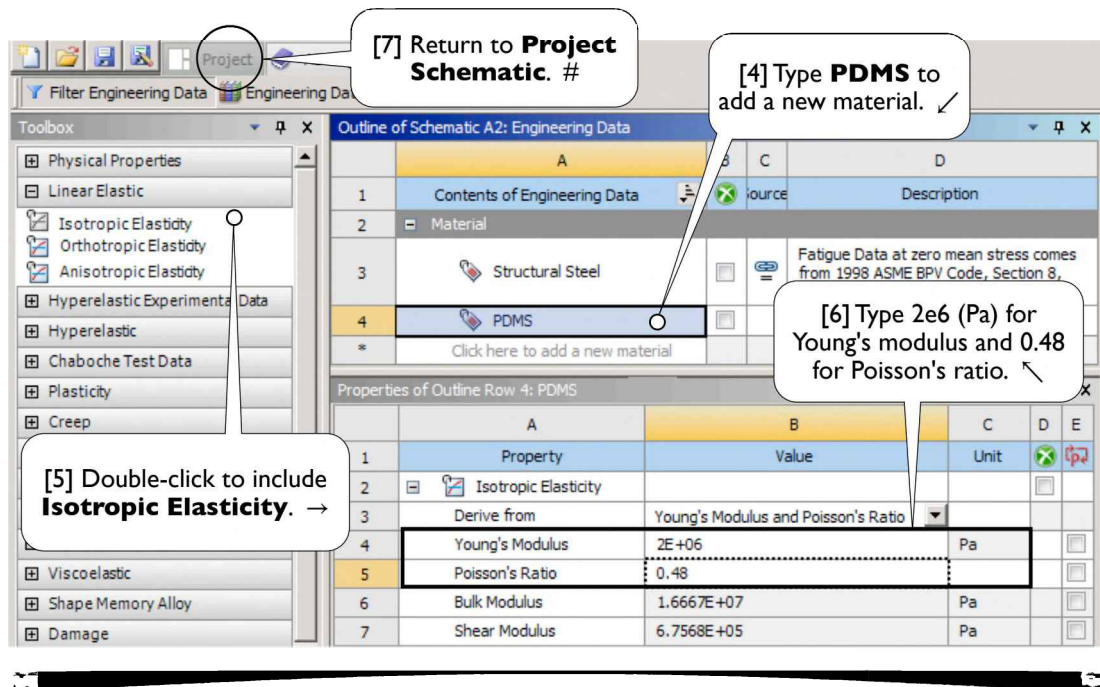
9.1.1 About the Pneumatic Fingers

[1] Simulation of the pneumatic finger was previewed in Section 1.1. In this section, we will walk through each step. Besides the information in 1.1.1 (pages 10-11), geometric details are given in the figure below. Due to the symmetry of the cross section [2], we model only half of the finger. The **mm-kg-N-s** unit system is used in this section. ↓



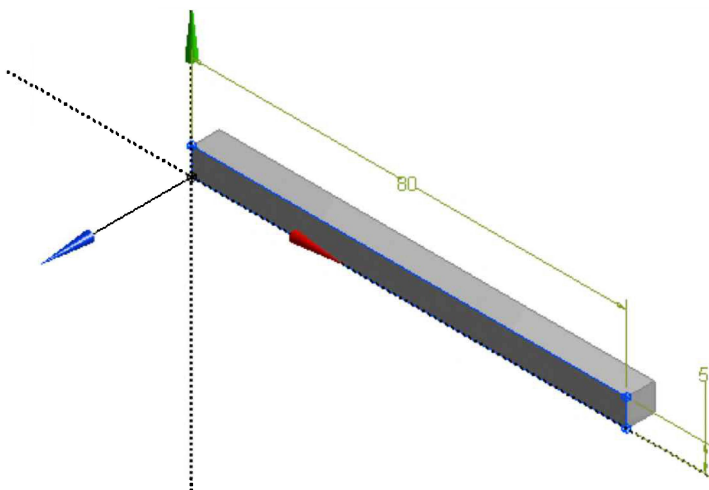
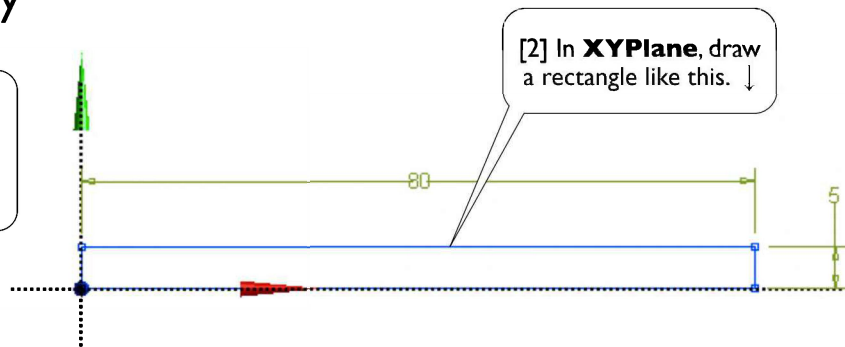
9.1.2 Start Up and Prepare Material Properties





9.1.3 Create Geometry

[1] Start up DesignModeler. Select **Millimeter** as the length unit and create a geometric model as shown in [2-7]. →

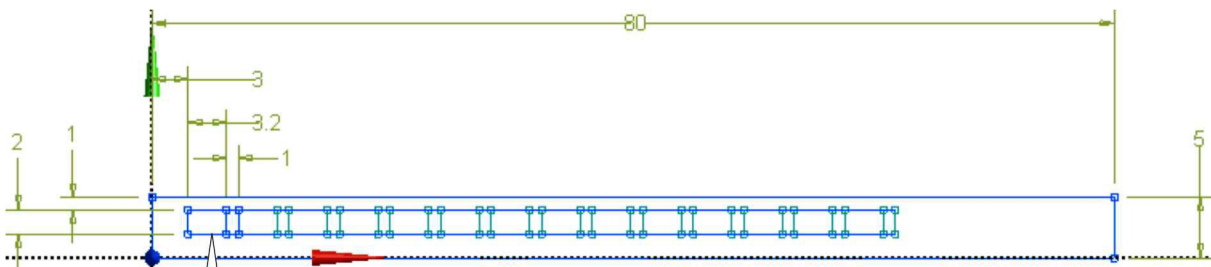


Details View		
Details of Extrude1		
Extrude	Extrude1	
Geometry	Sketch1	
Operation	Add Material	
Direction Vector	None (Normal)	
Direction	Reversed	
Extent Type	Fixed	
FD1, Depth (>0)	5.1 mm	
As Thin/Surface?	No	
Merge Topology?	Yes	
Geometry Selection: 1		
Sketch	Sketch1	

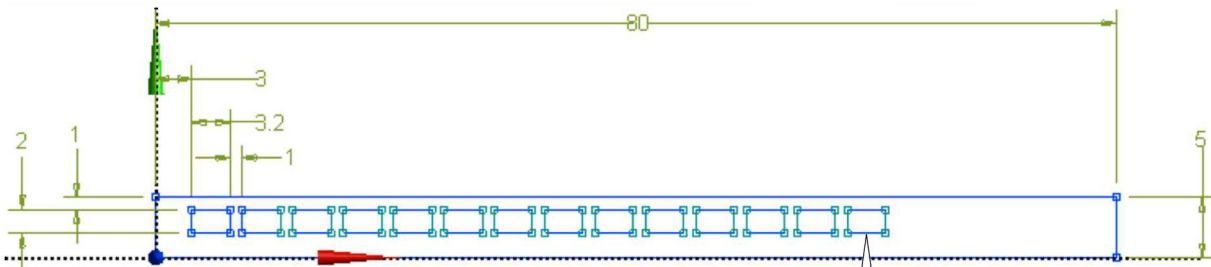
[3] **Extrude** to create a 3D block. ↙



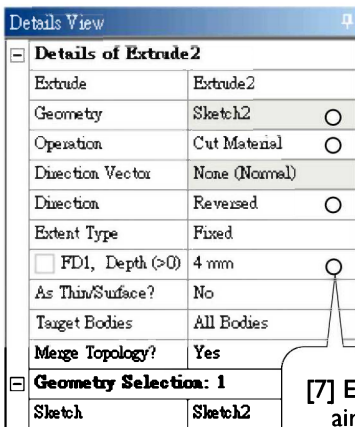
[4] On **XYPlane**, create a new sketch (**Sketch2**). Draw a rectangle and a vertical line like this. ↓



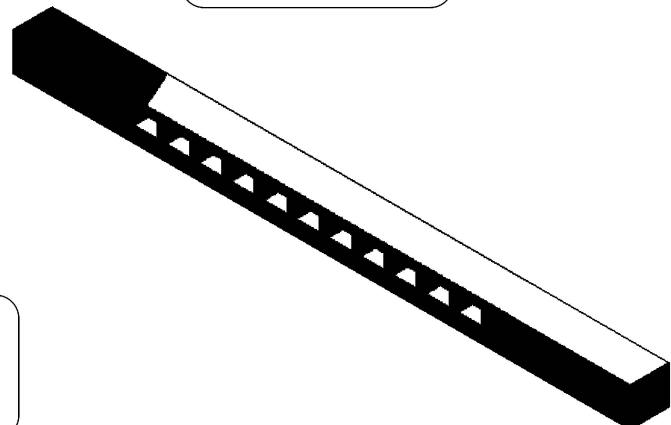
[5] Replicate 13 times, selecting all segments in the sketch except the leftmost vertical segment, using the upper-left corner as the paste handle. You may need to turn on **Selection Filter: Points**. ↓



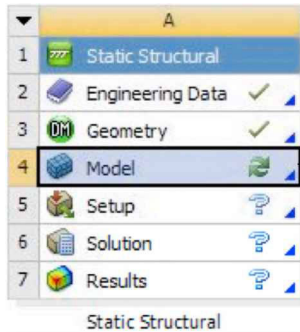
[6] Trim away unwanted segments. ↓



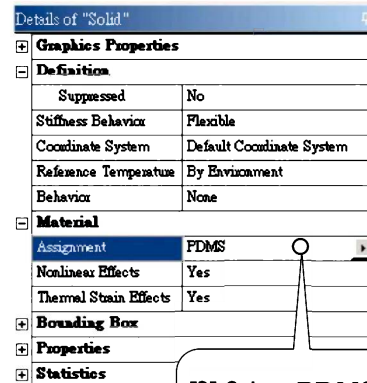
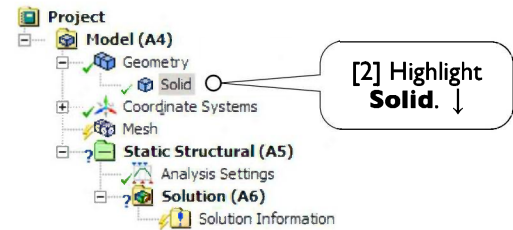
[7] Extrude-cut to create air chambers. Close DesignModeler. #



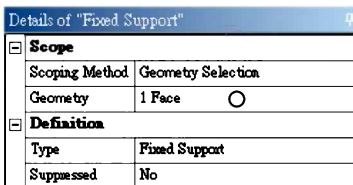
9.1.4 Assign Material



[1] Start up **Mechanical**.
Select **mm-kg-N-s** unit
system. →

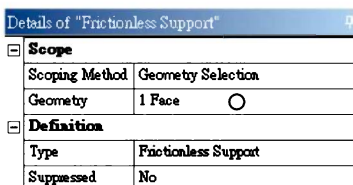


9.1.5 Set Up Environment Conditions



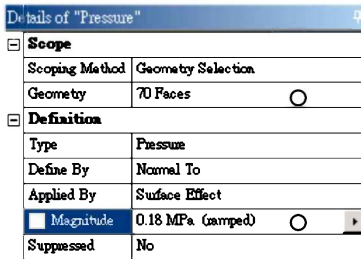
Fixed Support

[1] Set up a **Fixed Support** on this face. ↘



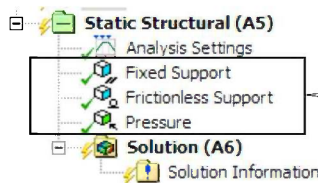
Frictionless Support

[2] Set up a **Frictionless Support** (i.e., plane of symmetry) on this face. ↙



Pressure: 0.18 MPa

[3] Apply a pressure of 0.18 MPa on the inner faces of the air chambers (70 faces in total). To efficiently select the 70 faces, you may select all faces (i.e., right-click-select **Select All**) and then deselect (using control-click) unwanted faces. ✓

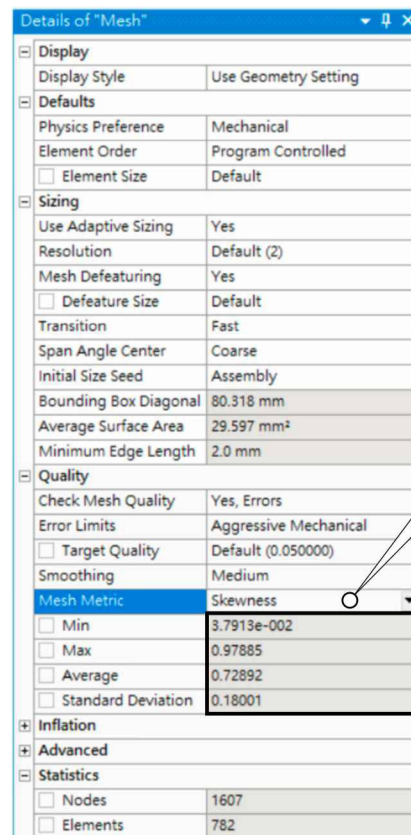
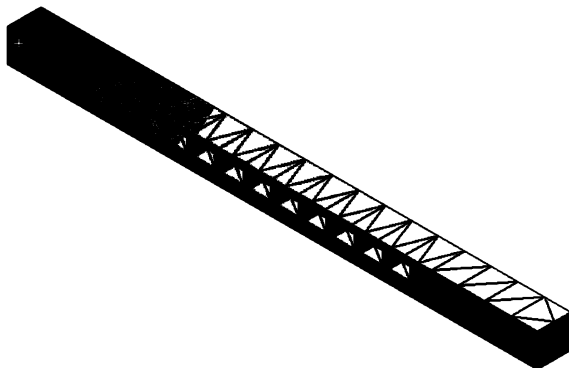


[4] The environment conditions should look like this. #

9.1.6 Mesh with Default Settings



[1] Generate mesh. →




[2] Select **Skewness** for **Mesh Metric**. ↓

[3] The default settings result in a poor mesh quality (see [4]). ↓

Skewness^[Refs 2, 3]

[4] Skewness, a measure of mesh quality, can be calculated for each element according to its geometry. Definition of skewness can be found in the on-line documentation^[Refs 2, 3]. For now, all you need to know is that it is a value ranging from 0 to 1, the smaller the better, and, as a guideline, element skewness should not be larger than 0.95. #

9.1.7 Improve Mesh Quality


Generate

[2, 5, 8] Generate mesh. ↙ ↓ ↓

[1] Select 4 for **Resolution**. →

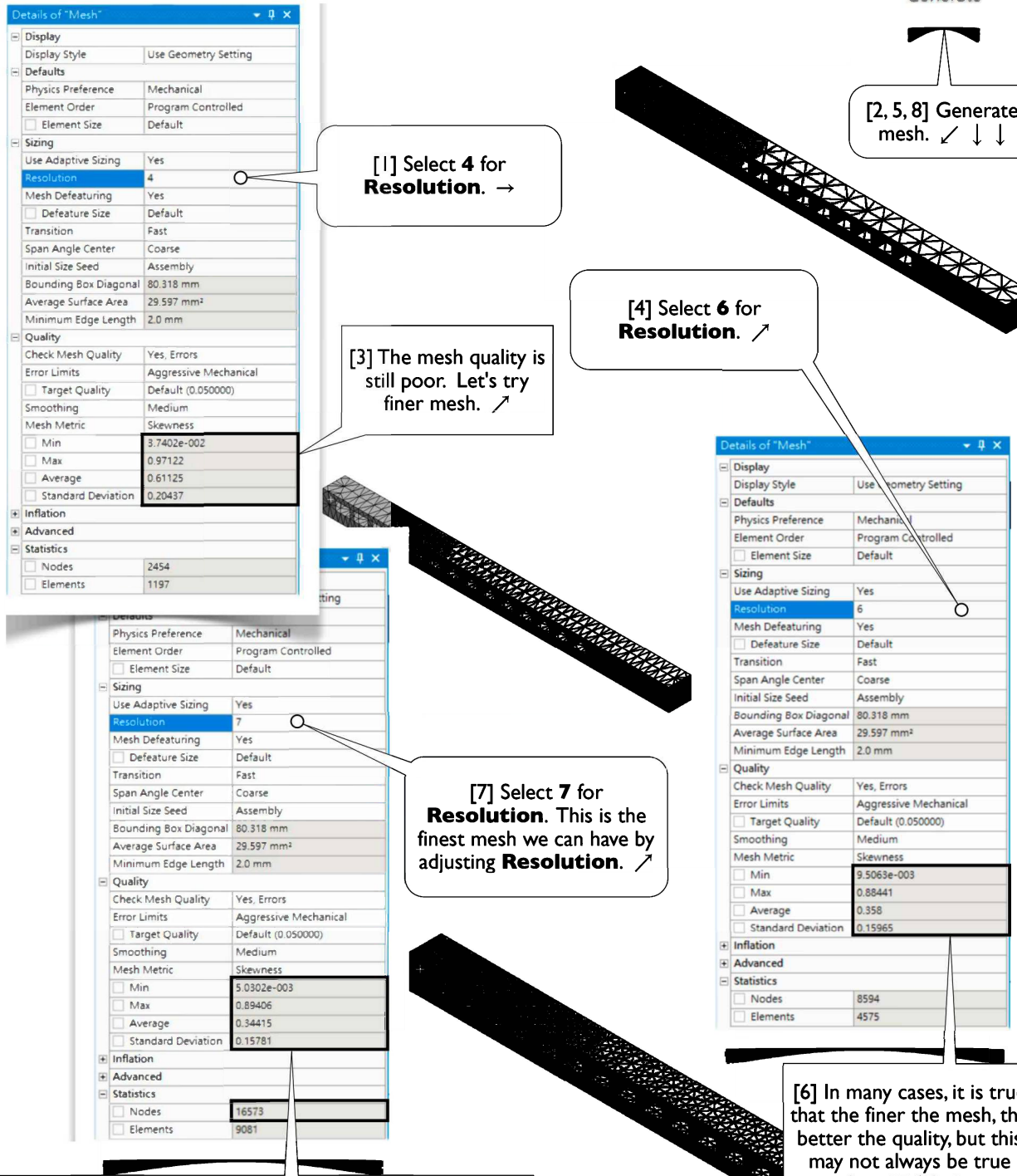
[3] The mesh quality is still poor. Let's try finer mesh. ↗

[4] Select 6 for **Resolution**. ↗

[7] Select 7 for **Resolution**. This is the finest mesh we can have by adjusting **Resolution**. ↗

[6] In many cases, it is true that the finer the mesh, the better the quality, but this may not always be true (e.g., [7-9]). ↖

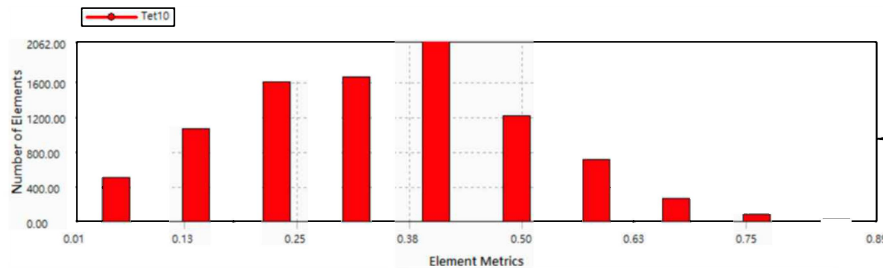
[9] A bar chart (see [10], next page) shows the distribution of the skewness. We'll use this mesh to obtain a solution. Note that the number of nodes here is 16573. ↙



Category	Value
Display	Display Style: Use Geometry Setting
Defaults	Physics Preference: Mechanical Element Order: Program Controlled Element Size: Default
Sizing	Use Adaptive Sizing: Yes Resolution: 4 Mesh Defeaturing: Yes Defeature Size: Default Transition: Fast Span Angle Center: Coarse Initial Size Seed: Assembly Bounding Box Diagonal: 80.318 mm Average Surface Area: 29.597 mm² Minimum Edge Length: 2.0 mm
Quality	Check Mesh Quality: Yes, Errors Error Limits: Aggressive Mechanical Target Quality: Default (0.050000) Smoothing: Medium Mesh Metric: Skewness Min: 3.7402e-002 Max: 0.97122 Average: 0.61125 Standard Deviation: 0.20437
Inflation	
Advanced	
Statistics	Nodes: 2454 Elements: 1197

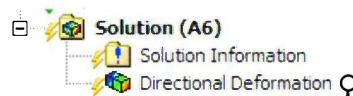
Category	Value
Display	Display Style: Use Geometry Setting
Defaults	Physics Preference: Mechanical Element Order: Program Controlled Element Size: Default
Sizing	Use Adaptive Sizing: Yes Resolution: 6 Mesh Defeaturing: Yes Defeature Size: Default Transition: Fast Span Angle Center: Coarse Initial Size Seed: Assembly Bounding Box Diagonal: 80.318 mm Average Surface Area: 29.597 mm² Minimum Edge Length: 2.0 mm
Quality	Check Mesh Quality: Yes, Errors Error Limits: Aggressive Mechanical Target Quality: Default (0.050000) Smoothing: Medium Mesh Metric: Skewness Min: 9.5063e-003 Max: 0.88441 Average: 0.358 Standard Deviation: 0.15965
Inflation	
Advanced	
Statistics	Nodes: 8594 Elements: 4575

Category	Value
Display	Display Style: Use Geometry Setting
Defaults	Physics Preference: Mechanical Element Order: Program Controlled Element Size: Default
Sizing	Use Adaptive Sizing: Yes Resolution: 7 Mesh Defeaturing: Yes Defeature Size: Default Transition: Fast Span Angle Center: Coarse Initial Size Seed: Assembly Bounding Box Diagonal: 80.318 mm Average Surface Area: 29.597 mm² Minimum Edge Length: 2.0 mm
Quality	Check Mesh Quality: Yes, Errors Error Limits: Aggressive Mechanical Target Quality: Default (0.050000) Smoothing: Medium Mesh Metric: Skewness Min: 5.0302e-003 Max: 0.89406 Average: 0.34415 Standard Deviation: 0.15781
Inflation	
Advanced	
Statistics	Nodes: 16573 Elements: 9081



[10] With **Mesh** highlighted, a bar chart shows the distribution of skewness. #

9.1.8 Set Up Solution Branch



[1] Insert a **Directional Deformation**. →

Details of "Directional Deformation"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Directional Deformation
Orientation	Y Axis
By	Time
Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No

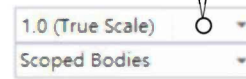
[2] In this study, we focus on the Y-directional deformation. #

9.1.9 Obtain a Linear Solution

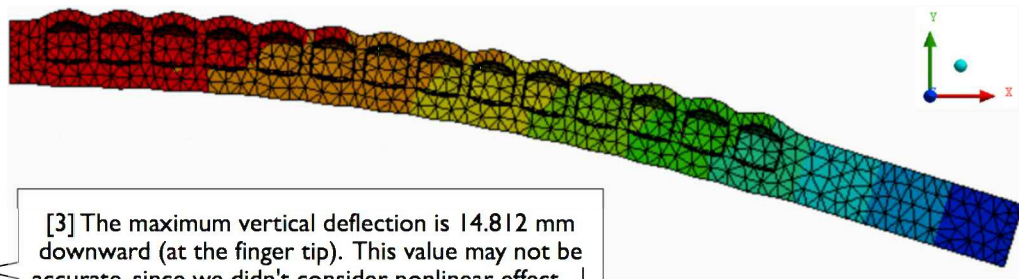
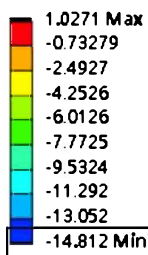


[1] Solve. It takes only a few seconds to complete the linear solution. →

[2] Select **True Scale**. ↓



A: Static Structural
Directional Deformation
Type: Directional Deformation(Y Axis)
Unit: mm
Global Coordinate System
Time: 1



[3] The maximum vertical deflection is 14.812 mm downward (at the finger tip). This value may not be accurate, since we didn't consider nonlinear effect. ↓

Obtain a Linear Solution before Nonlinear Simulations

[4] It is a good practice to make sure a linear solution can be obtained before a nonlinear simulation is performed. A linear simulation takes much less computational time than a nonlinear one. Nonlinearity should be considered in this case, since the deflection is large. The linear solution, however, provides a way of model checking. #

9.1.10 Obtain a Nonlinear Solution

[1] Highlight **Analysis Settings**. ↓

[2] Turn on **Auto Time Stepping**. ↓

[3] Select **Time**. ↓

[4] Type 0.05 (s) for **Initial Time Step**, 0.01 (s) for **Minimum Time Step**, and 0.05 (s) for **Maximum Time Step**. ↓

[5] Turn on **Large Deflection**. The simulation will include geometric nonlinearity. We will explain these settings further in Chapter 13. ↗

[6] Highlight **Solution Information**. ↓

[7] And select **Displacement Convergence**. ↓

[8] Solve the model. It takes a while to complete the nonlinear solution, depending on your hardware capability. ↓

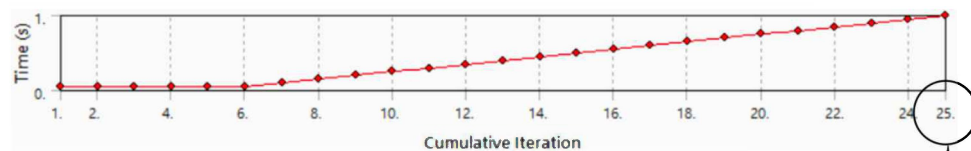
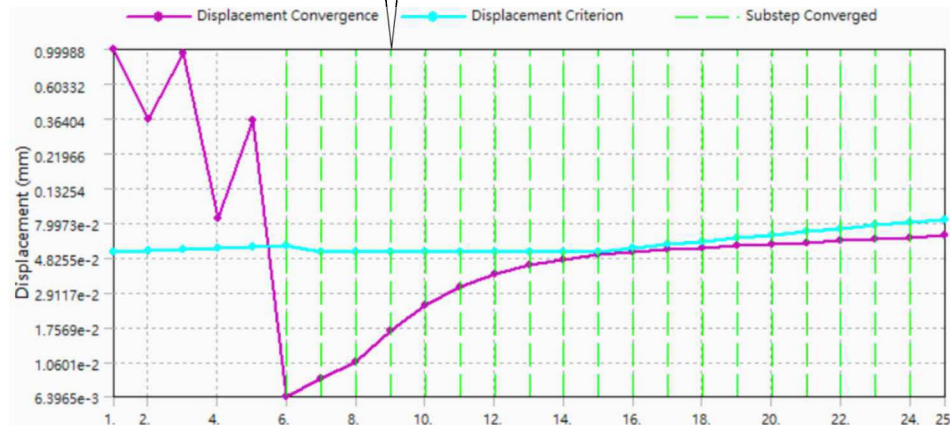
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	On
Define By	Time
Initial Time Step	5.e-002 s
Minimum Time Step	1.e-002 s
Maximum Time Step	5.e-002 s
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Restart Controls	
Nonlinear Controls	
Output Controls	
Analysis Data Management	
Visibility	

Solution Information	
Solution Output	Displacement Convergence
Newton-Raphson Residuals	0
Identify Element Violations	0
Update Interval	2.5 s
Display Points	All
FE Connection Visibility	
Activate Visibility	Yes
Display	All FE Connectors
Draw Connections Attached To	All Nodes
Line Color	Connection Type
Visible on Results	No
Line Thickness	Single
Display Type	Lines

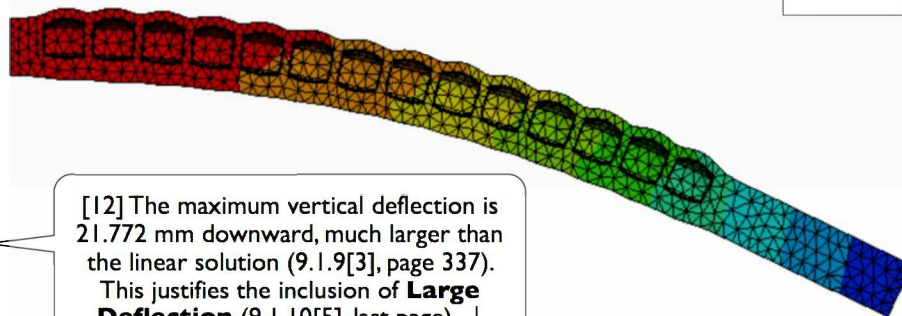
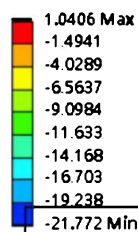
Displacement Convergence

[9] Each substep of a nonlinear simulation involves an iterative process. Force and displacement values are used as convergence criteria. These concepts will be further explained in 13.1.5 (page 473). ↵

[11] Interpretation of these curves will be discussed in Chapter 13. ↓



A: Static Structural
 Directional Deformation
 Type: Directional Deformation(Y Axis)
 Unit: mm
 Global Coordinate System
 Time: 1



[12] The maximum vertical deflection is 21.772 mm downward, much larger than the linear solution (9.1.9[3], page 337). This justifies the inclusion of **Large Deflection** (9.1.10[5], last page). ↓

[10] The solution converges in 25 iterations. ↑

Element Shapes

[13] In many cases, nonlinear simulations can be challenging. Meshing quality plays an important role in the convergence of nonlinear solution. The mesh metric (e.g., skewness; see 9.1.7[9], page 336) is a measure of mesh quality. Skewness often can be improved by refining elements. It sometimes needs a large number of elements to achieve a mesh quality that is good enough to make the solution converge. In other cases, it may never achieve an acceptable mesh quality by simply refining elements.

Another factor affecting convergence is the shapes of elements. In general, hexahedra are more efficient than tetrahedra (see 9.3.13 and 9.3.14, page 361). In the following exercises, let's try to mesh the model with hexahedra. #

9.1.11 Mesh with **Hex Dominant** Method

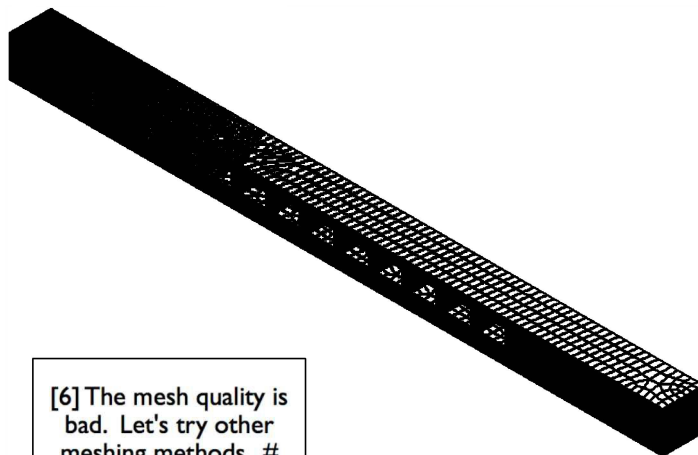
[1] With **Mesh** highlighted, click **Method** to insert a mesh method. →

[2] Select the solid body. ↓

[3] Select **Hex Dominant** for **Method**. ↓

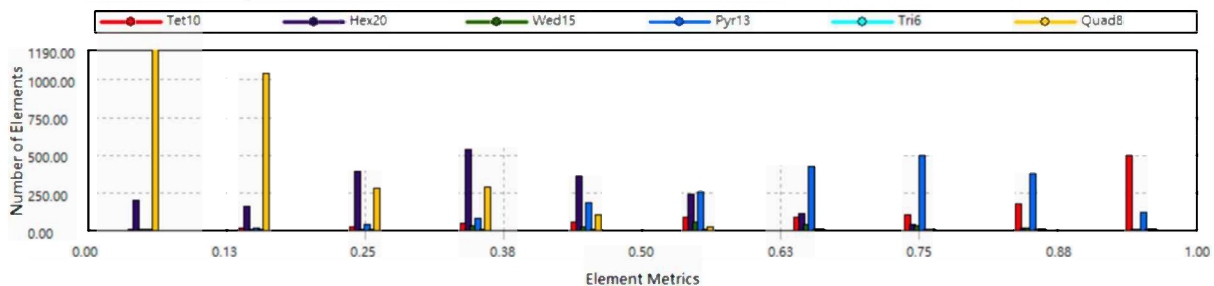
[4] It is a warning. Click to read the message. It says that this geometry may not be suitable for **Hex Dominant** method. Ignore the message. ✓

[5] Generate mesh. ↓

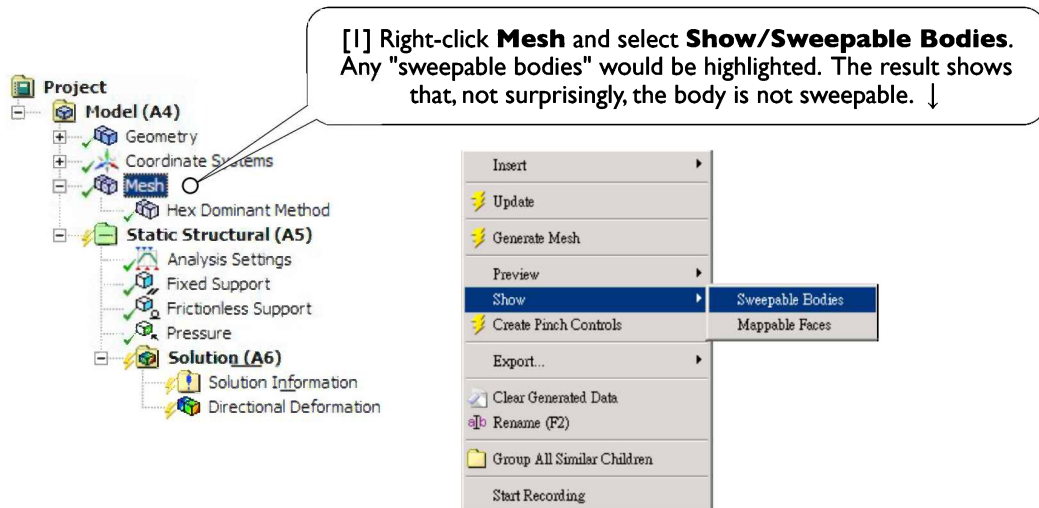


Details of "Mesh"

Display	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
<input type="checkbox"/> Element Size	Default
Sizing	
Use Adaptive Sizing	Yes
Resolution	7
Mesh Defeaturing	Yes
<input type="checkbox"/> Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Diagonal	80.318 mm
Average Surface Area	29.597 mm ²
Minimum Edge Length	2.0 mm
Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	Skewness
<input type="checkbox"/> Min	1.3057e-010
<input type="checkbox"/> Max	0.99999
<input type="checkbox"/> Average	0.41312
<input type="checkbox"/> Standard Deviation	0.30237
Inflation	
Advanced	
Statistics	
<input type="checkbox"/> Nodes	18058
<input type="checkbox"/> Elements	7952



9.1.12 Mesh with **Sweep** Method



Sweepable Bodies

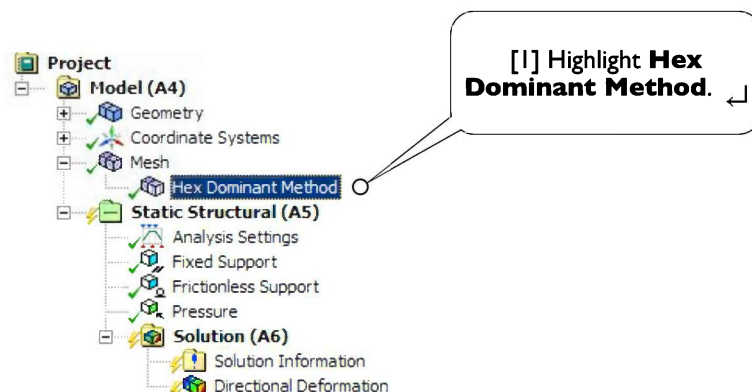
[2] As mentioned in 5.3.2[2] (page 228), a simple idea for generating hexahedral elements is to mesh a face (or faces) of a body with quadrilaterals and then "sweep" along its depth direction to the other end face (or faces) of the body. The starting faces are called the *source faces* and the ending faces are called the *target faces*. The source or target faces can be either manually or automatically selected.

Not all bodies are sweepable. In our case, there is only one body, and it is not sweepable.

Mesh with **MultiZone** Method

For non-sweepable bodies, Workbench provides a sophisticated method of generating hexahedral elements, called **MultiZone** method. The idea of **MultiZone** method is to divide a non-sweepable body into several sweepable bodies, and then apply **Sweep** method on each of the bodies. Actually, we already applied this method to mesh the beam bracket model in 5.1.13[1-5] (page 217). #

9.1.13 Mesh with **MultiZone** Method



Details of "MultiZone" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Not Allowed
Element Midside Nodes	Use Global Setting
Size/Tag Selection	Automatic
Source Scoping Method	Program Controlled
Source	Program Controlled
Sweep Size Behavior	Sweep Element Size
Sweep Element Size	Default
Advanced	
Preserve Boundaries	Protected
Mesh Based Defeaturing	Off
Minimum Edge Length	2. mm
Write ICFM CFD Files	No

[2] Select **MultiZone**. →



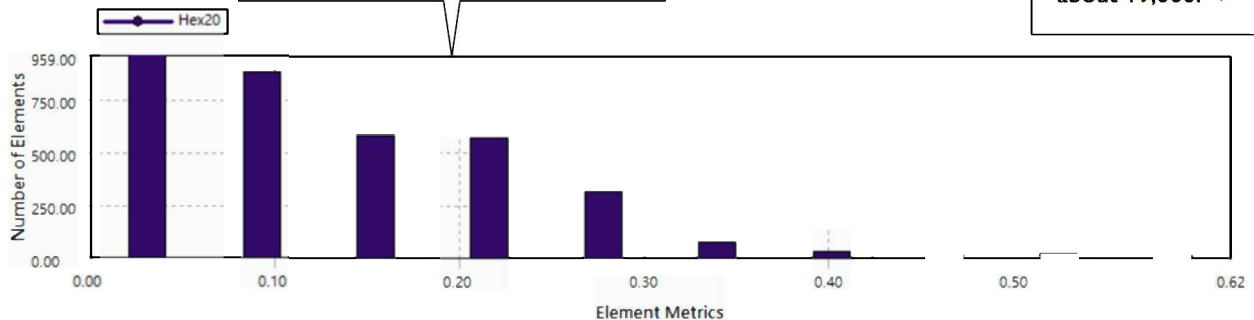
[3] Generate mesh. ✓

Details of "Mesh"	
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	Default
Sizing	
Use Adaptive Sizing	Yes
Resolution	7
Mesh Defeaturing	Yes
Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Diagonal	80.318 mm
Average Surface Area	29.597 mm²
Minimum Edge Length	2.0 mm
Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
Target Quality	Default (0.050000)
Smoother	Medium
Mesh Metric	Skewness
Min	1.3058e-010
Max	0.61747
Average	0.12717
Standard Deviation	0.10271
Inflation	
Advanced	
Statistics	
Nodes	19201
Elements	3395

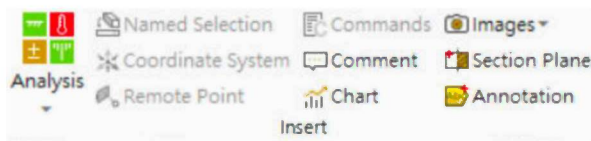
[4] Workbench successfully meshes the body with hexahedral elements. ↘

[6] With **Mesh** highlighted, a bar chart shows the distribution of skewness. Note that the mesh contains hexahedral elements only. #

[5] The mesh quality is good. We'll use this mesh to obtain a solution. Note that the number of nodes is about 19,000. ←



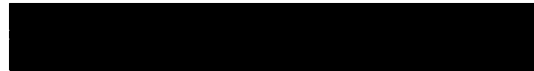
9.1.14 Examine Mesh Using Section View



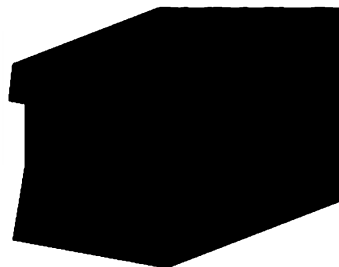
[1] With **Mesh** highlighted, select **Home/Insert/Section Plane** in the toolbar to create a section plane. ↓



[2] Draw a line (it is called a section plane) like this. ↓

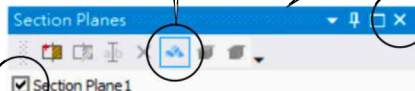


[5] Rotate the view to see the internal mesh. ↓



[4] Click **Show Whole Elements**. ↑

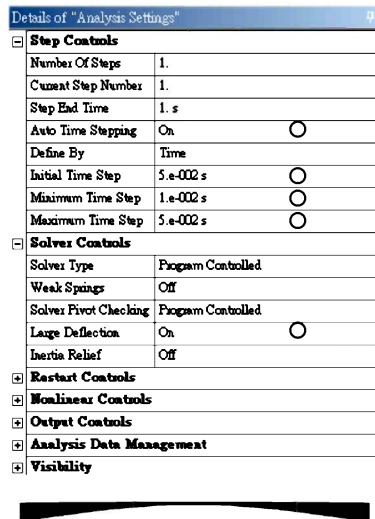
[3] A **Section Planes** window on the left-bottom lists the section planes you've created. ←



[7] Click here to close the window pane. You may reopen this window by select **Home/Insert/Section Plane** again. #

[6] Click to deactivate (uncheck) the section view and return to full view. →

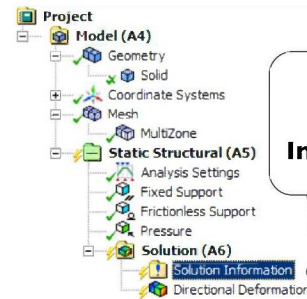
9.1.15 Obtain a Nonlinear Solution



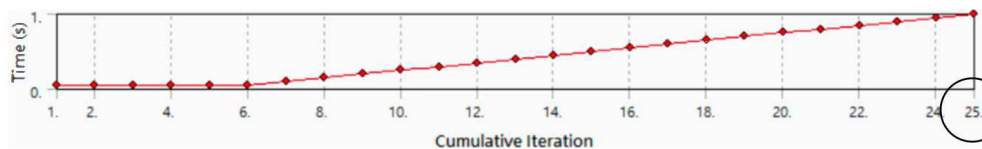
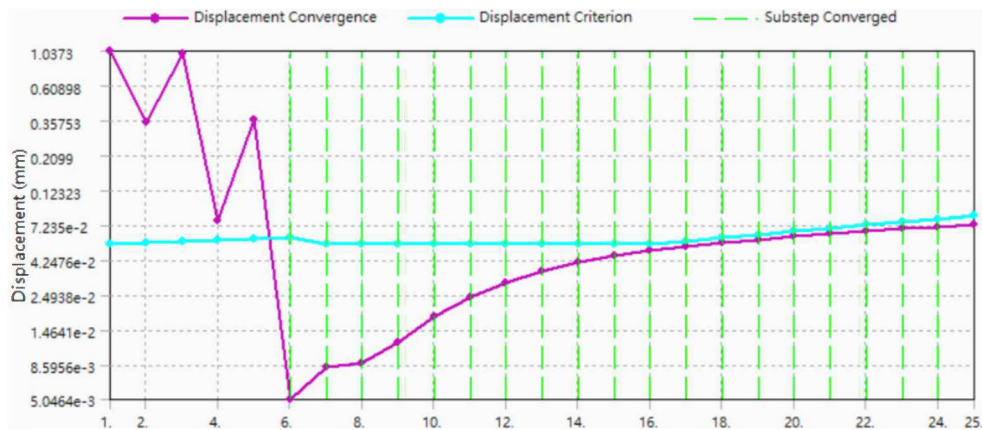
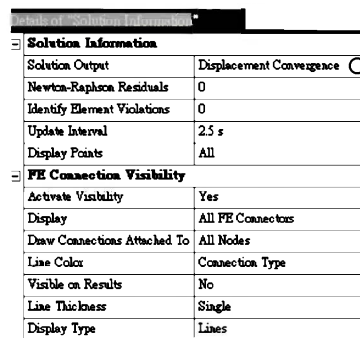
[1] We're using the same analysis settings. →



[3] Solve. ↓

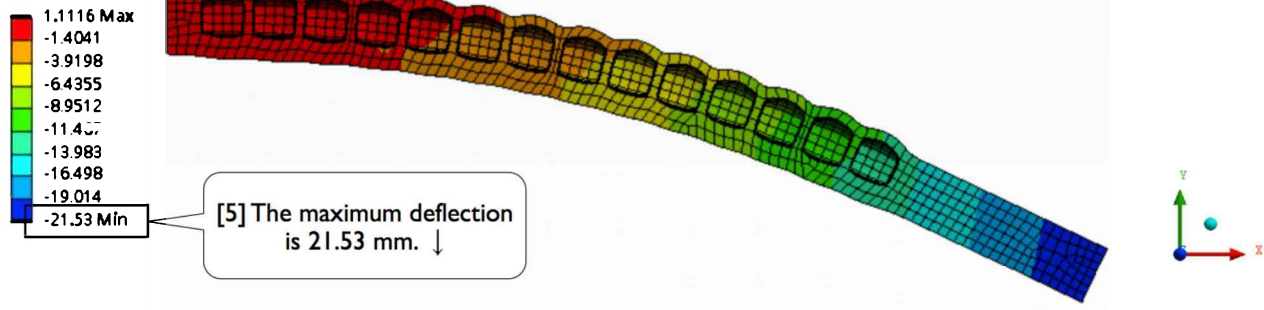


[2] Highlight Solution Information. ✓



[4] In my computer, the solution converges in 25 iterations. ↩

A: Static Structural
 Directional Deformation
 Type: Directional Deformation(Y Axis)
 Unit: mm
 Global Coordinate System
 Time: 1



Wrap Up

[6] Save the project and exit Workbench. ↓

Remark

[7] As mentioned in 1.1.8[8] (page 19), when assuming a linear material, we are also assuming the compressive behavior is the same as tensile behavior, but this is usually not true for an elastomer under such a large deformation. (Note that the upper portion of the finger is subject to tension, while the lower portion is subject to compression.)

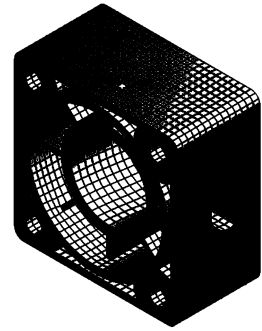
Hyperelasticity, a more accurate material model for elastomer under large deformation, will be introduced in PART C of Section 14.1 (pages 532-535). #

References

1. This exercise is adapted from an unpublished work led by Prof. Chao-Chieh Lan of the Department of Mechanical Engineering, NCKU.
2. All Help>Meshing>Meshing User's Guide>Global Mesh Controls>Quality Group>Mesh Metric
3. All Help>Mechanical APDL>Theory Reference>12.1. Element Shape Testing

Section 9.2

Cover of Pressure Cylinder

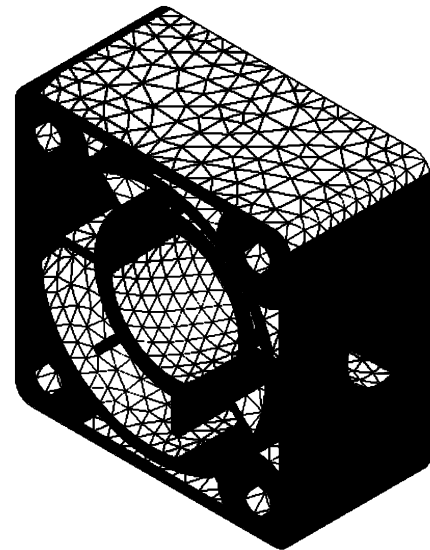


9.2.1 About the Cylinder Cover

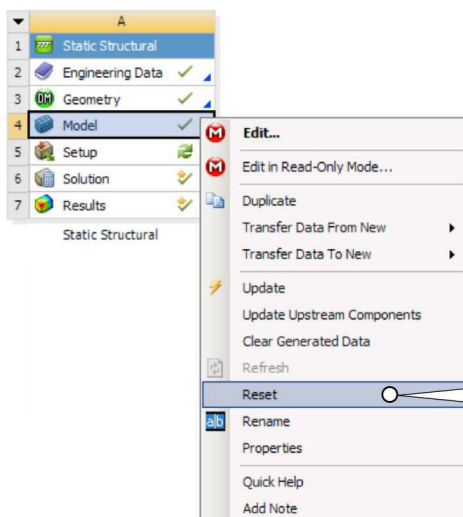
[1] In this section, we will use the cylinder cover (Sections 4.2 and 5.2) to demonstrate some additional meshing techniques.

The geometry of the cover is relatively complicated. It seems that a tetrahedral mesh is the only feasible method. There is nothing wrong with a tetrahedral mesh as long as the mesh quality is good enough. Examining the mesh generated in 5.2.5[3-5] (page 221), we see that the mesh quality is bad (9.2.3[1-3], next page). The mesh quality needs to be improved. The simplest way is to adjust the relevance values. That sometimes works, although increasing the problem size, but sometimes fails. For a linear static simulation, problem size seems no big deal, but for a nonlinear or dynamic simulation, the problem size should be kept as small as possible, to maintain an acceptable computing time.

Note also that the purpose of this section is to demonstrate meshing techniques, rather than finding the best mesh for the cylinder cover. #

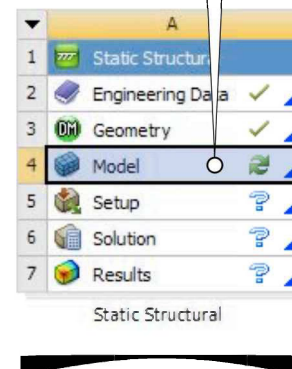


9.2.2 Open the Project **Cover**

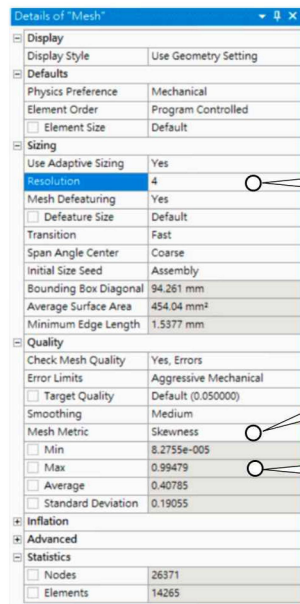


[1] Launch Workbench and open the project **Cover**, which was saved in Section 5.2. Right-click **Model** and select **Reset** (to delete the generated data). →

[2] Double-click **Model** to start up **Mechanical**. Make sure the unit system is **mm-k-g-N-s. #**



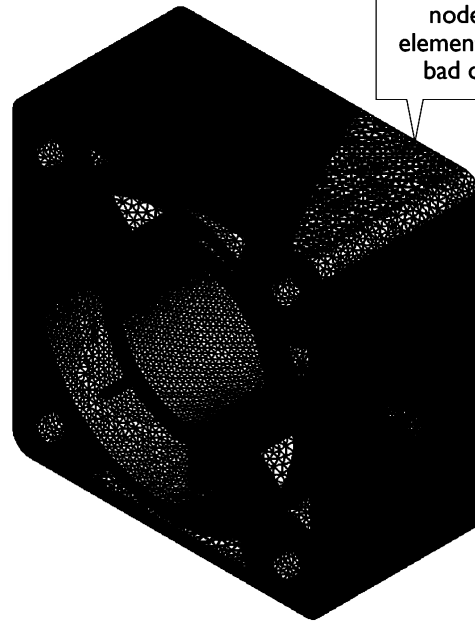
9.2.3 Increase Mesh Density Using Global Mesh Controls



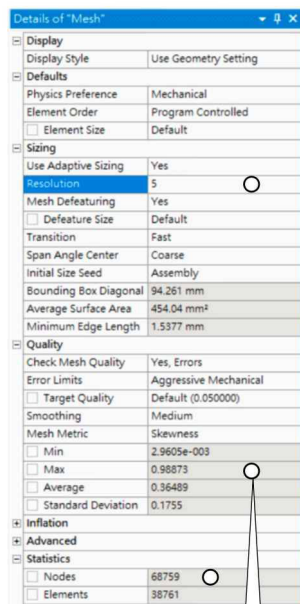
[1] Highlight **Mesh** and select **4** for **Resolution**. Generate the mesh. This is the mesh used in 5.2.5[3-5], page 221. ↓

[2] Select **Skewness** for **Mesh Metric**. ↓

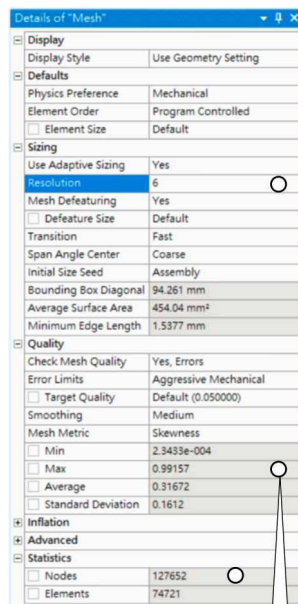
[3] Some elements have large skewness. ✓



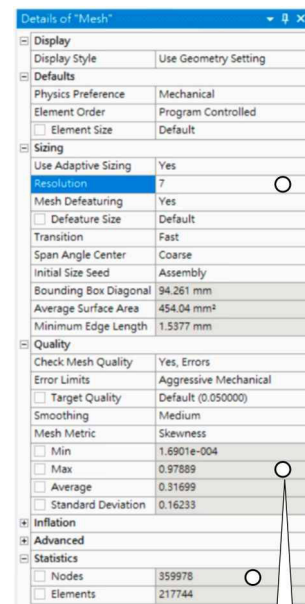
[7] With a mesh of more than 360,000 nodes, some elements still have bad quality. ↓



[4] As we increase the mesh density by setting new relevance values, the **Max** remains high. →



[5] Even when the number of nodes increases to more than 100,000, the **Max** remains high. →




[6] In this case, increasing mesh density has little effect on **Max** value. ↑


Increasing Mesh Density Is Not a Panacea

[8] The lesson we learned here is that increasing mesh density, although often reducing average skewness, is not a universal remedy for eliminating large skewness. We need to learn other meshing techniques. #

9.2.4 Mesh with **Patch Conforming** Method



[1] With **Mesh** highlighted, select **Method** to insert a mesh method. ↓



[6] Generate mesh. ↓

[2] Select the solid body. ↓

Details of "Patch Conforming Method" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Tetrahedrons
Algorithm	Patch Conforming
Element Midside Nodes	Use Global Setting

[3] Select **Tetrahedrons**. ↓

[4] **Patch Conforming** is the default **Algorithm**. ↗

[5] Highlight **Mesh** and select **4** for **Resolution** and generate the mesh. For the rest of this section, we always use **4** for **Resolution**. ←

Details of "Mesh"	
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
<input type="checkbox"/> Element Size	Default
Sizing	
Use Adaptive Sizing	Yes
Resolution	4
Mesh Defeaturing	Yes
<input type="checkbox"/> Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Diagonal	94.261 mm
Average Surface Area	454.04 mm²
Minimum Edge Length	1.5377 mm
Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	Skewness
<input type="checkbox"/> Min	8.2755e-005
<input type="checkbox"/> Max	0.99479
<input type="checkbox"/> Average	0.40785
<input type="checkbox"/> Standard Deviation	0.19055
Inflation	
Advanced	
Statistics	
<input type="checkbox"/> Nodes	26371
<input type="checkbox"/> Elements	14265

[7] The mesh is the same as that in 9.2.3[4] (last page). In fact, **Patch Conforming** method is the default mesh method for a complicated geometry. ↓

Patch Conforming and **Patch Independent** Methods

[8] The faces of a solid body are also called the *patches*. The basic idea of **Patch Conforming** is to mesh all the faces of the body with triangles and then "grow" inward to create tetrahedra. In this way, the exterior shape of the body (i.e., shapes of its faces) is respected (preserved), thus the name **Patch Conforming**. For complicated geometry, this is the default method.

On the other hand, **Patch Independent** creates tetrahedra from inside out. The outermost nodes are then projected onto the boundary faces and the element edges are created. In this way, the mesh's exterior shape may deviate from the original geometry, thus the name **Patch Independent**.

In some cases, when too many details exist that cause meshing difficulty, we may resort to **Patch Independent** algorithm and ignore these details. However, it is your responsibility to make sure that ignoring those details wouldn't distort the geometry too much. #

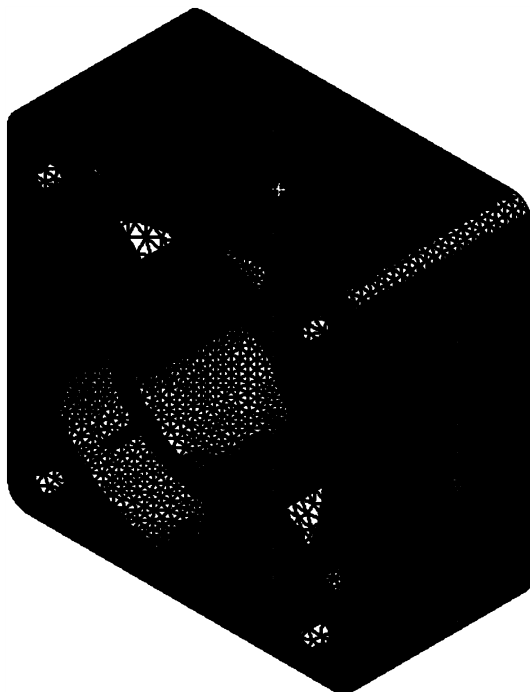
9.2.5 Mesh with **Patch Independent** Method

Details of "Patch Independent" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Tetrahedras
Algorithm	Patch Independent
Element Order	Use Global Setting
Advanced	
Defined By	Max Element Size
Max Element Size	Default
Feature Angle	30.0°
Mesh Based Defeaturing	Off
Refinement	Proximity and Curvature
Min Size Limit	2.5 mm
Num Cells Across Gap	Default
Curvature Normal Angle	Default
Smooth Transition	Off
Growth Rate	Default
Minimum Edge Length	1.5377 mm
Write ICEM CFD Files	No

[2] Select **Patch Independent**. ✓

[4] We decide to use 2.5 (mm) for **Min Size Limit**. ↘

[3] Since the minimum edge length is reportedly 1.5377... ↑



[1] Highlight **Patch Conforming Method**. ←



Details of "Mesh"	
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
<input type="checkbox"/> Element Size	Default
Sizing	
Use Adaptive Sizing	Yes
Resolution	4
Mesh Defeaturing	Yes
<input type="checkbox"/> Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Diagonal	94.261 mm
Average Surface Area	454.04 mm²
Minimum Edge Length	1.5377 mm
Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	Skewness
<input type="checkbox"/> Min	1.2452e-003
<input type="checkbox"/> Max	0.68501
<input type="checkbox"/> Average	0.23975
<input type="checkbox"/> Standard Deviation	0.15097
Inflation	
Advanced	
Statistics	
<input type="checkbox"/> Nodes	365787
<input type="checkbox"/> Elements	240206

[5] **Generate Mesh**. It may take a while. The mesh quality is okay... ↓

[6] But the mesh size becomes large. ↙

Details of "Patch Independent" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Tetrahedrons
Algorithm	Patch Independent
Element Order	Use Global Setting
Advanced	
Defined By	Max Element Size
Max Element Size	Default
Feature Angle	30.0 °
Mesh Based Defeaturing	Off
Refinement	Proximity and Curvature
Min Size Limit	3.5 mm
Num Cells Across Gap	Default
Curvature Normal Angle	Default
Smooth Transition	Off
Growth Rate	Default
Minimum Edge Length	1.5377 mm
Write ICEM CFD Files	No

[7] If we use a coarser element size... →

Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	Skewness
Min	5.2229e-003
Max	0.90092
Average	0.42696
Standard Deviation	0.19004
Inflation	
Advanced	
Statistics	
Nodes	37756
Elements	21895

[8] The mesh size does reduce substantially. ✓

Details of "Patch Independent" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Tetrahedrons
Algorithm	Patch Independent
Element Midside Nodes	Use Global Setting
Advanced	
Defined By	Max Element Size
Max Element Size	Default
Feature Angle	30.0 °
Mesh Based Defeaturing	Off
Refinement	Proximity and Curvature
Min Size Limit	6 mm
Num Cells Across Gap	Default
Curvature Normal Angle	Default
Smooth Transition	Off
Growth Rate	Default
Minimum Edge Length	1.5377 mm
Write ICEM CFD Files	No

[9] If we further increase the element size... →

Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	Skewness
Min	2.0704e-002
Max	0.95501
Average	0.57351
Standard Deviation	0.20153

[10] It may worsen the mesh quality... ↓

[11] And may generate a distorted "patch," not conforming to the geometry (i.e., "patch" is independent of the geometry). #

9.2.6 Mesh with **Hex Dominant** Method

Details of "Hex Dominant Method" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Hex Dominant
Element Midside Nodes	Use Global Setting
Free Face Mesh Type	Quad/Tri
Control Messages	Yes, Click To Display...



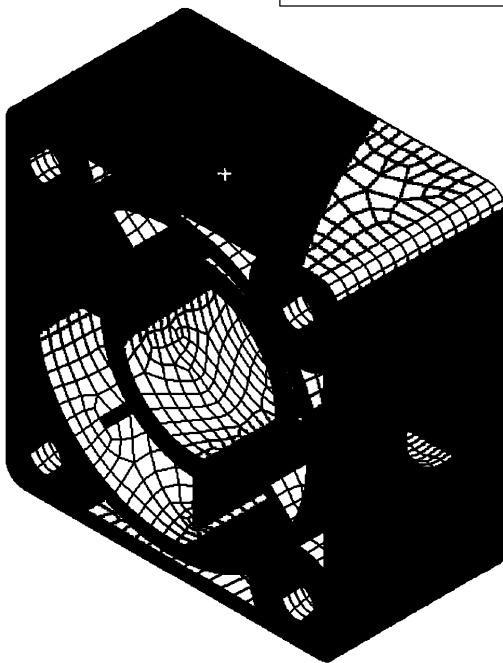
[1] Highlight **Patch Independent**. ←

[2] Select **Hex Dominant** method. ↓



[4] **Generate Mesh**.
It may take a while. ✓

[3] Ignore the warning. The warning says that **Hex Dominant** method for this case, where the ratio of volume to surface area is low, may result in poor mesh quality. →



[5] The mesh appears to be fine... →

[6] But the quality is poor... ↓

Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanic
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	Skewness
<input type="checkbox"/> Min	1.3057e-010
<input type="checkbox"/> Max	1.
<input type="checkbox"/> Average	0.51442
<input type="checkbox"/> Standard Deviation	0.30344
Inflation	
Advanced	
Statistics	
<input type="checkbox"/> Nodes	39857
<input type="checkbox"/> Elements	19313

[7] And the mesh count is high. ↓

Hex Dominant Method

[8] An idea of **Hex Dominant** is to mesh the body with **Patch Conforming** first and then combine tetrahedra to form hexahedra: two tetrahedra form a hexahedron. It usually leaves some tetrahedra that cannot be combined to form hexahedra, thus the name **Hex Dominant**. After forming hexahedra, the algorithm tries to adjust the nodes to improve the mesh quality further.

Note that, **Hex Dominant** method, by its nature, is a method of patch conforming; that is, the faces are not distorted. In fact, all methods except **Patch Independent** are patch conforming. #

9.2.7 Mesh with **MultiZone** Method

Details of "MultiZone" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Hexa Dominant
Element Midside Nodes	Use Global Setting
Size/Tag Selection	Automatic
Source Scoping Method	Program Controlled
Source	Program Controlled
Sweep Size Behavior	Sweep Element Size
Sweep Element Size	Default
Advanced	
Preserve Boundaries	Protected
Mesh Based Defeaturing	Off
Minimum Edge Length	1.5377 mm
Write ICEM CFD Files	No

Quality

Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	Skewness
Min	1.3058e-010
Max	1.
Average	0.52958
Standard Deviation	0.25395

[1] Highlight **Hex Dominant Method**. ←

[2] Select **MultiZone** method. →

[3] **Generate Mesh**. It may take a while. ✓

[4] The mesh appears perfect... ↘

[5] But the quality is actually not good. Some internal elements have poor quality. ↓

MultiZone Method

[6] As mentioned, the idea of **MultiZone** method is to decompose a non-sweepable body into several sweepable bodies, and then apply **Sweep** method on each body. The selection of source faces can be automatic or manual. In this case, you may try to select source faces manually.

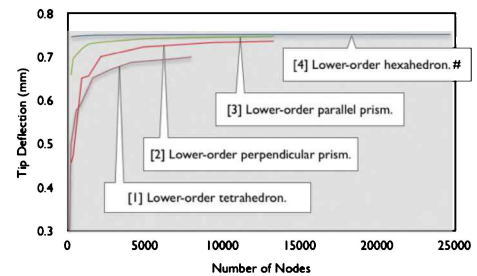
Workbench is often smart enough to decompose the body into sweepable bodies. But, since these bodies are an integral part, the boundaries between the decomposed bodies must be conformal (i.e, the boundaries must have the same surface mesh); these constraints may complicate the meshing task. ↓

Wrap Up

[7] Save the project and exit Workbench. #

Section 9.3

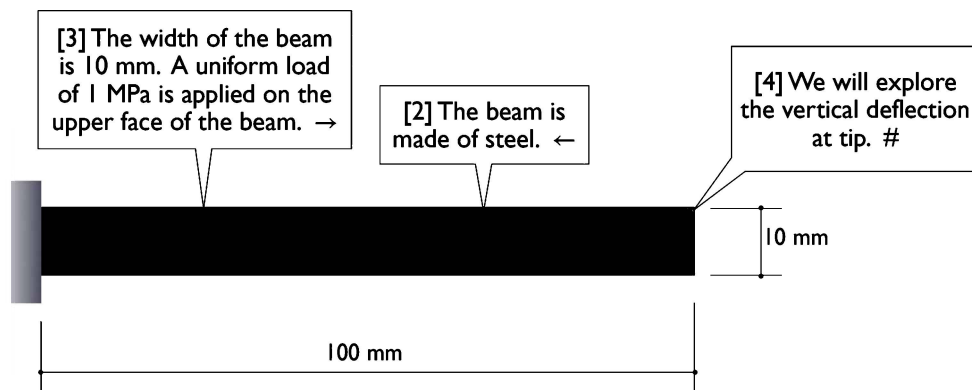
Convergence Study of 3D Solid Elements



The main purpose of this section is to study 3D solid elements convergence behavior. A secondary purpose is to serve as an exercise for mesh controls techniques. A cantilever beam of rectangular cross section is used for these purposes. The conclusions drawn from the convergence study are crucial for CAE engineers. This section can be viewed as a sequel of PART C of Section 3.5 (pages 161-163).

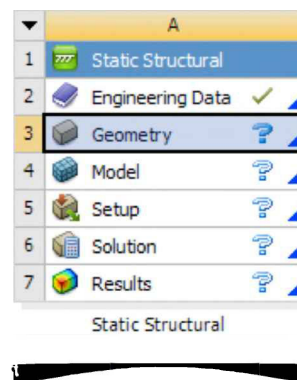
9.3.1 About the Cantilever Beam

[1] The cantilever beam is made of steel and of size 100 mm x 10 mm x 10 mm; a uniform load of 1 MPa is applied on the upper face of the beam [2-4]. Convergence of three solid element shapes will be compared, namely hexahedron, prism, and tetrahedron (1.3.3[2-3, 5], page 38). ↘



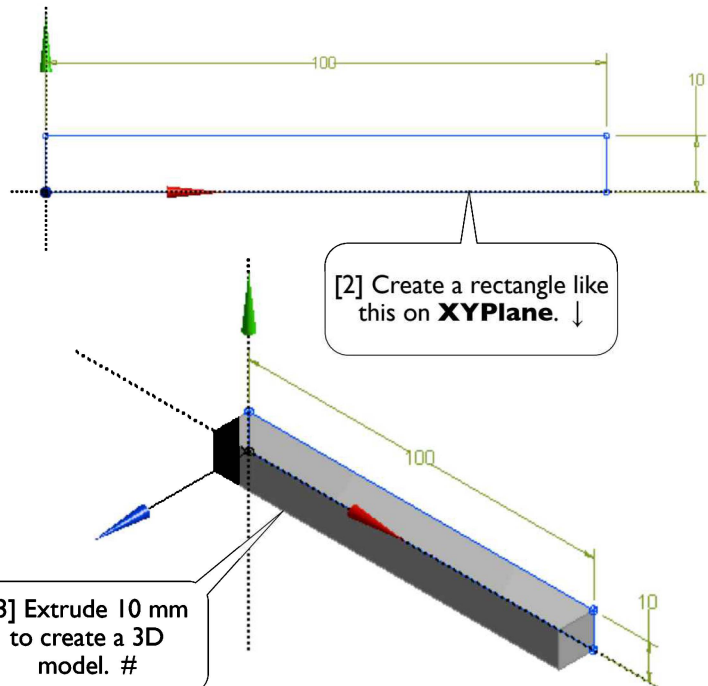
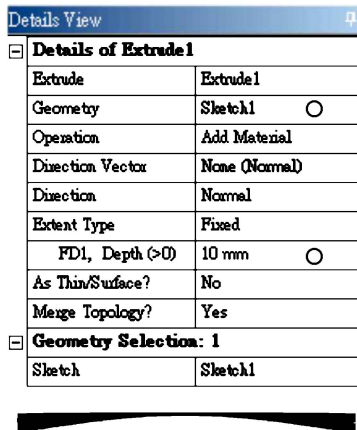
9.3.2 Start Up a New Project

[1] Launch Workbench. Create a **Static Structural** system. Save the project as **Cantilever**. Start up DesignModeler. Select **Millimeter** as the length unit. #



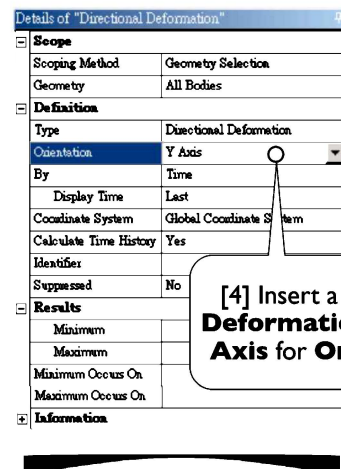
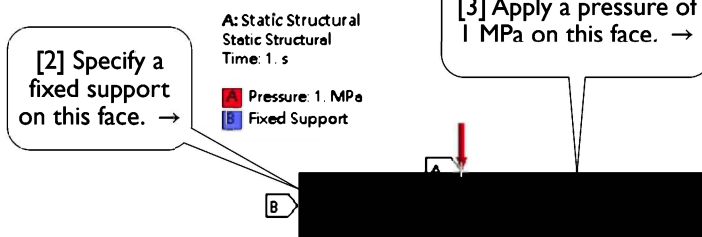
9.3.3 Create a 3D Model in DesignModeler

[1] On **XYPlane**, create a rectangle [2].
Extrude the sketch to create a 3D model [3].
Close DesignModeler. →



9.3.4 Set Up Support, Load, and Solution Objects

[1] Start up **Mechanical** and select the **mm-kg-N-s** unit system. Specify a **Fixed Support** on the left face [2]. Apply a pressure of 1 MPa on the upper face [3]. Insert a **Directional Deformation** under the solution branch and select **Y Axis** for **Orientation** [4]. ✓



[4] Insert a **Directional Deformation** and select **Y Axis** for **Orientation**. #

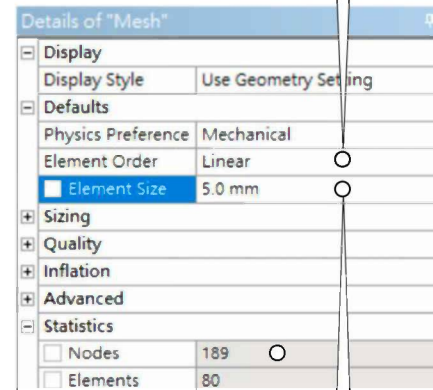
9.3.5 Lower-Order Hexahedra

[1] For a model of such a regular geometry, the default mesh control settings will generate an all-hexahedra mesh of higher-order elements [2]. To generate lower-order hexahedra, select **Linear** for **Element Order** [3]. For each run, change the element size [4]. Resulting tip deflections are recorded in the table below. The convergence curve is shown in [5]. →

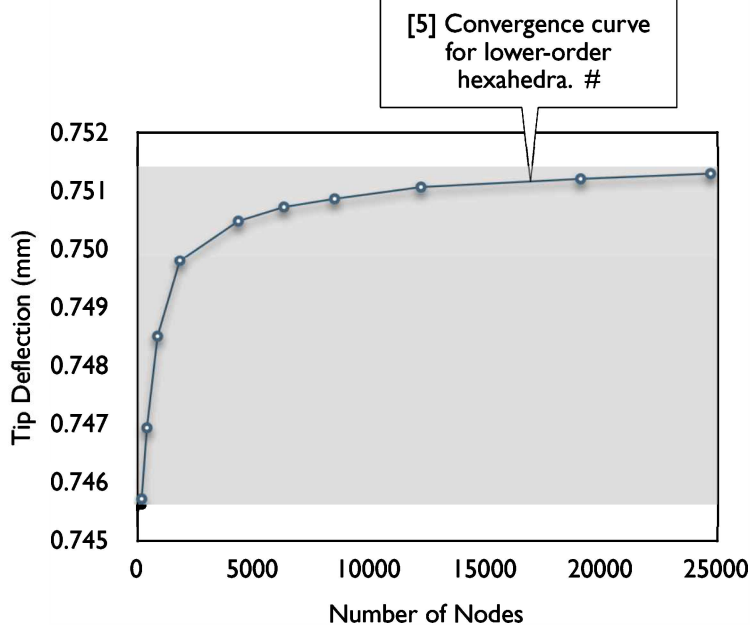
Element Size (mm)	Number of Nodes	Tip Deflection (mm)
5	189	0.74571
4	416	0.74693
3	875	0.74850
2	1836	0.74980
1.5	4352	0.75048
1.3	6318	0.75072
1.2	8500	0.75086
1	12221	0.75106
0.9	19097	0.75120
0.8	24696	0.75129

[2] The default mesh settings will generate an all-hexahedra mesh of higher-order elements. ↓

[3] Select **Linear** for **Element Order**. ↓



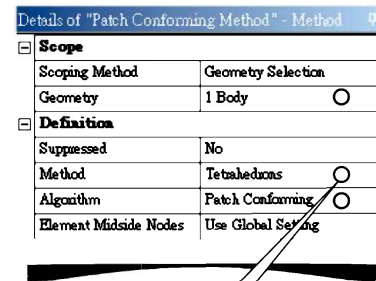
[4] Change **Element Size** for each run. ←



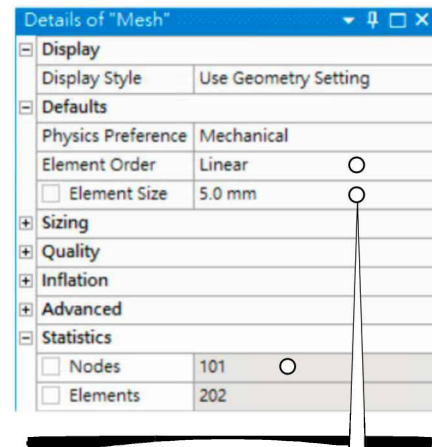
9.3.6 Lower-Order Tetrahedra

[1] Highlight **Mesh** in the project tree and click **Method** to insert a mesh control method. Select **Tetrahedrons** method [2]. Now, change element size for each run [3]. Resulting tip deflections are recorded in the table below. The convergence curve is shown in [4]. →

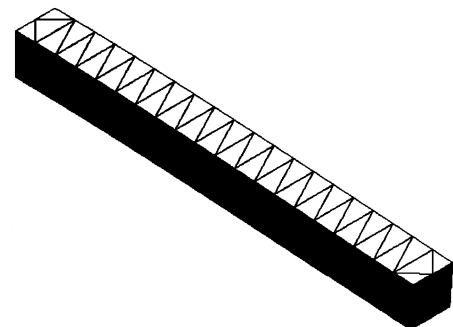
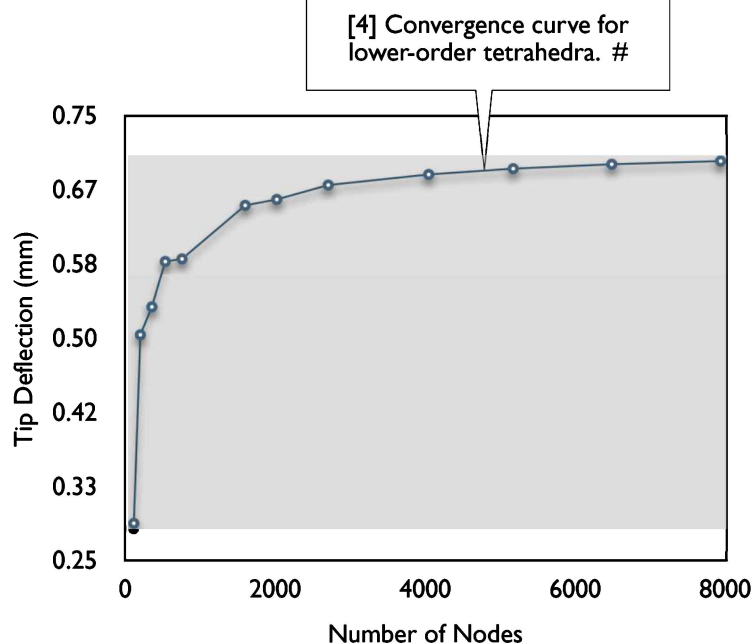
Element Size (mm)	Number of Nodes	Tip Deflection (mm)
5	101	0.29182
4	188	0.50308
2.5	341	0.53437
2	520	0.58585
1.5	743	0.58876
1.2	1584	0.64860
1	2005	0.65522
0.8	2690	0.67128
0.7	4028	0.68313
0.6	5154	0.68960
0.55	6466	0.69452
0.5	7916	0.69807



[2] Select **Tetrahedrons** method. ↓



[3] Change **Element Size** for each run. ←

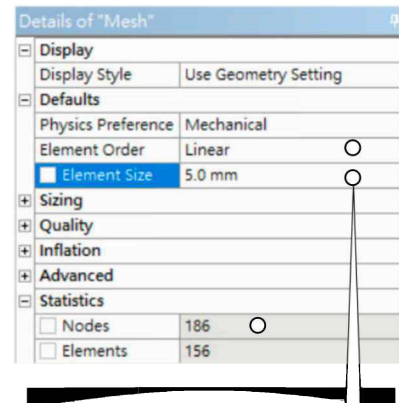
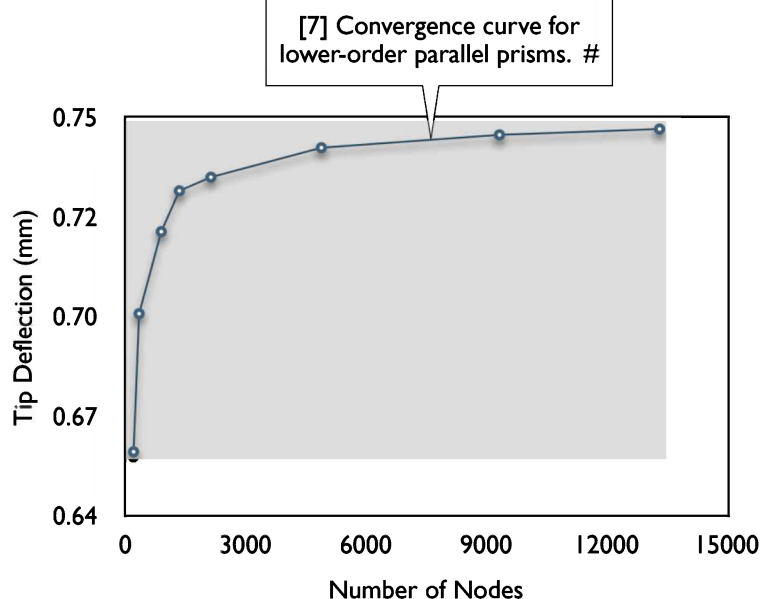
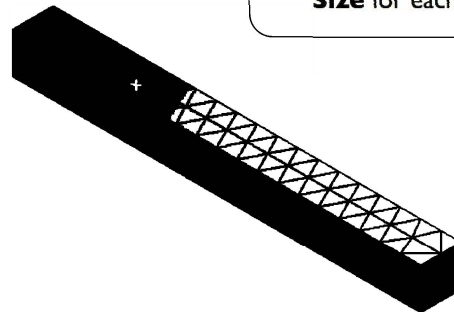
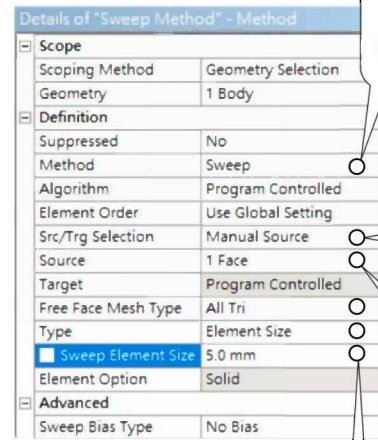


9.3.7 Lower-Order Prisms (Parallel to Loading Direction)

[1] Highlight **Patch Conforming Method**. In the details view, change to **Sweep** method [2]. Set up the source face [3-4]. For each run, change both sweep element size [5] and global element size [6]. Resulting tip deflections are recorded in the table below. The convergence curve is shown in [7].

Note that the prisms are oriented such that their heights are parallel to the loading (bending) direction. We will refer to the elements oriented in this way as "parallel prisms" for the rest of this section. We will show in 9.3.8 (next page) that the convergence curve will be different if the prisms are oriented differently. →

Element Size (mm)	Number of Nodes	Tip Deflection (mm)
5	186	0.65779
4	324	0.69584
3	875	0.71844
2.4	1326	0.72969
2	2112	0.73344
1.5	4864	0.74156
1.2	9300	0.74510
1	13288	0.74671



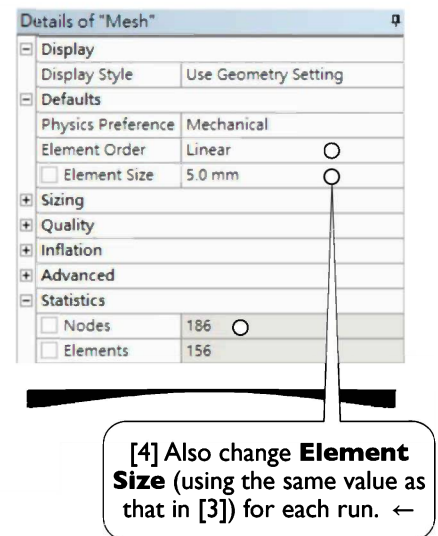
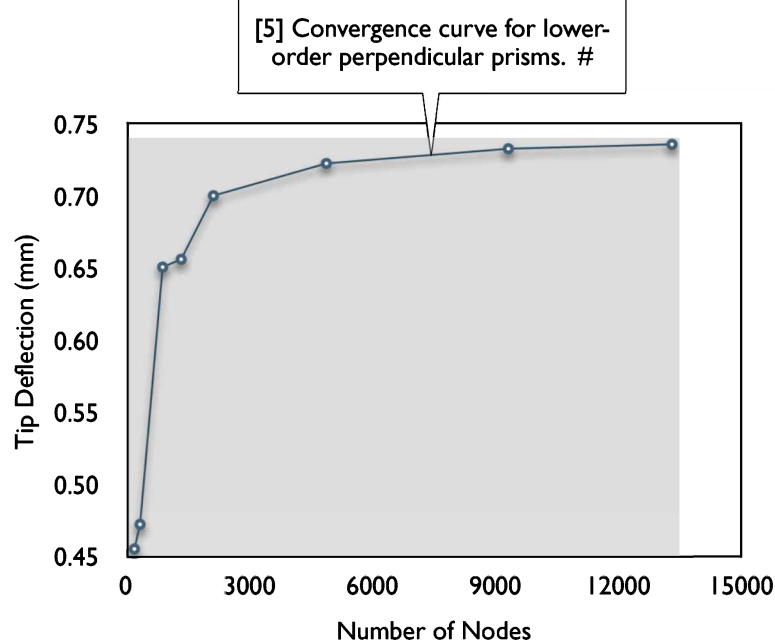
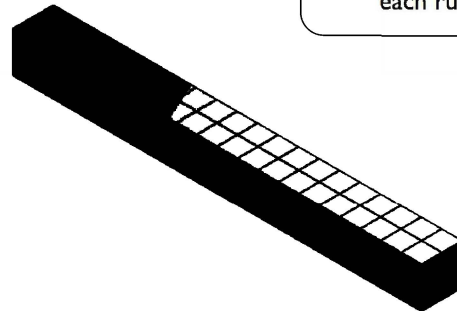
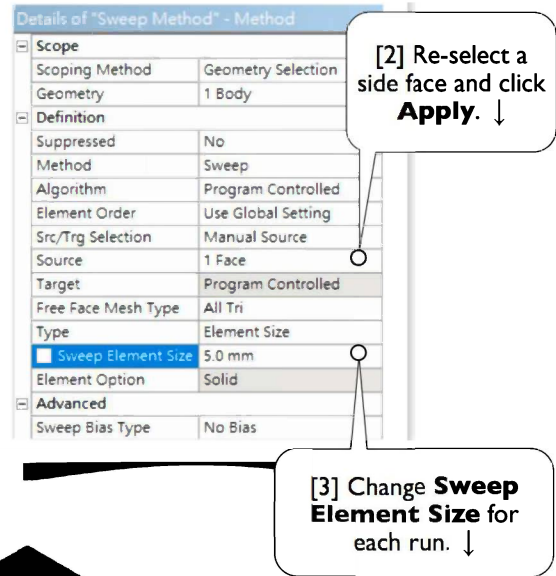
[6] Also change **Element Size** (using the same value as that in [5]) for each run. ←

9.3.8 Lower-Order Prisms (Perpendicular to Loading Direction)

[1] In the details view of **Sweep Method**, re-select the source face [2]. For each run, change both sweep element size [3] and global element size [4]. Resulting tip deflections are recorded in the table below. The convergence curve is shown in [5].

Note that the prisms are oriented such that their heights are perpendicular to the loading (bending) direction. We will refer to the elements oriented in this way as "perpendicular prisms" for the rest of this section. →

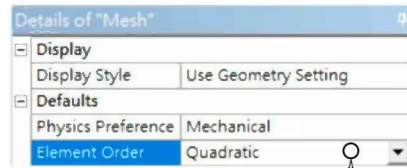
Element Size (mm)	Number of Nodes	Tip Deflection (mm)
5	186	0.45546
4	324	0.47229
3	875	0.65078
2.4	1326	0.65622
2	2112	0.70027
1.5	4864	0.72260
1.2	9300	0.73279
1	13288	0.73582



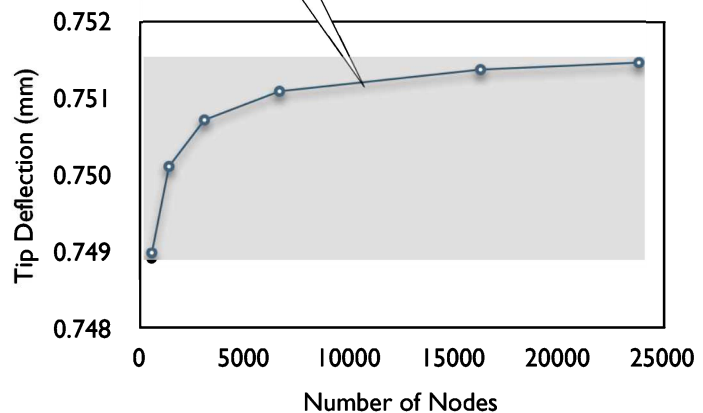
9.3.9 Higher-Order Hexahedra

[1] Delete **Sweep Method**. Repeat all the steps in 9.3.5 (page 355), using quadratic elements [2] and change the element sizes as shown below. →

Element Size (mm)	Number of Nodes	Tip Deflection (mm)
5	621	0.74899
4	1440	0.75011
3	3125	0.75071
2	6696	0.75108
1.5	16256	0.75136
1.3	23787	0.75145



[3] Convergence curve for higher-order hexahedra. #



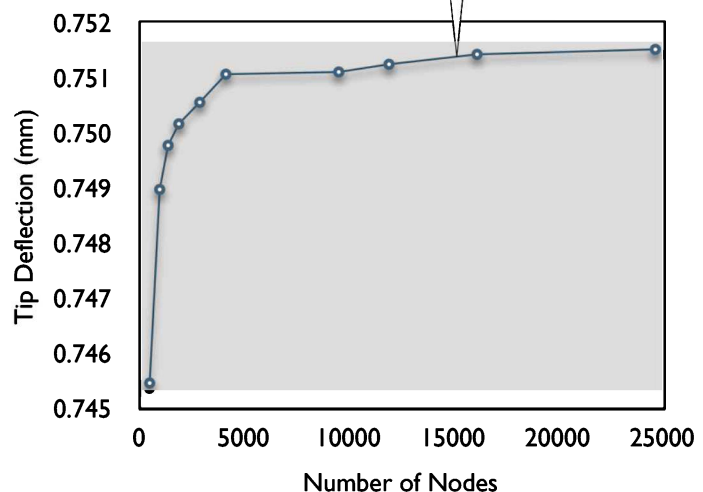
[4] Select **Quadratic** for **Element Order**. ←

9.3.10 Higher-Order Tetrahedra

[1] Repeat all the steps in 9.3.6 (page 356), using quadratic elements (9.3.9[2], this page) and change the element sizes as shown below. →

Element Size (mm)	Number of Nodes	Tip Deflection (mm)
5	493	0.74546
4	971	0.74897
3	1371	0.74977
2.5	1883	0.75016
2	2880	0.75055
1.5	4127	0.75106
1.2	9508	0.75110
1	11905	0.75124
0.8	16102	0.75142
0.7	24607	0.75151

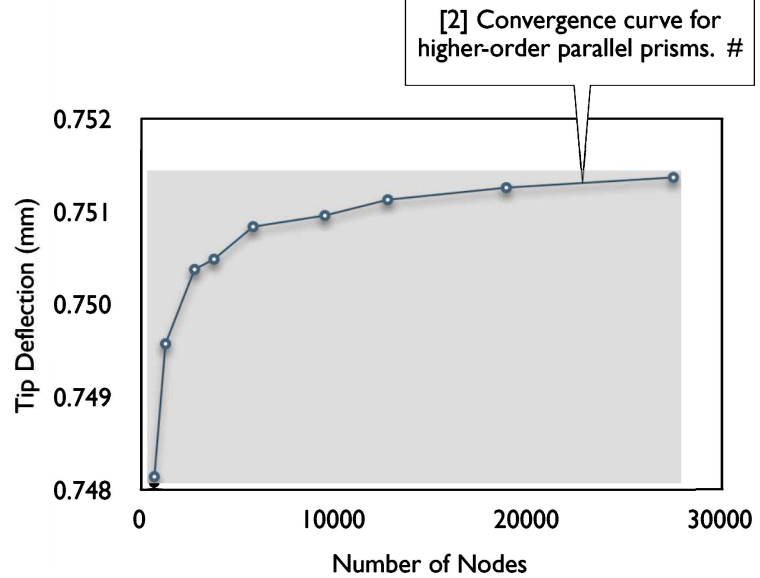
[2] Convergence curve for higher-order tetrahedra. #



9.3.11 Higher-Order Parallel Prisms

[1] Repeat all the steps in 9.3.7 (page 357), using quadratic elements (9.3.9[2], last page) and change the element sizes as shown below. →

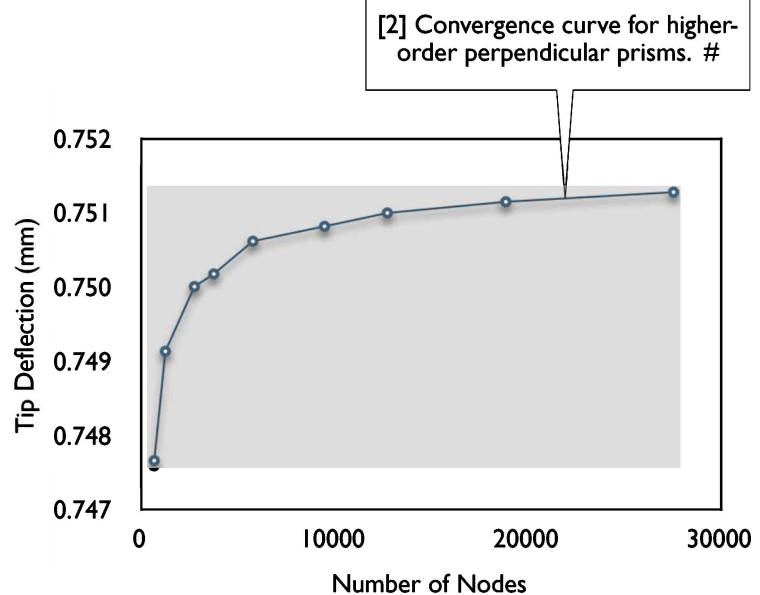
Element Size (mm)	Number of Nodes	Tip Deflection (mm)
5	727	0.74815
4	1303	0.74958
3.3	2803	0.75038
3	3805	0.75049
2.4	5827	0.75084
2	9530	0.75096
1.8	12779	0.75113
1.6	18902	0.75126
1.4	27529	0.75137



9.3.12 Higher-Order Perpendicular Prisms

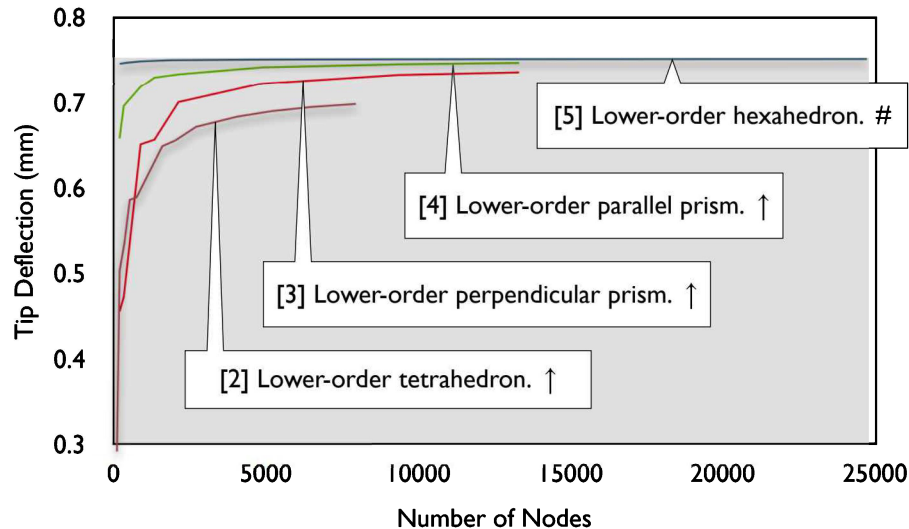
[1] Repeat all the steps in 9.3.8 (page 358), using quadratic elements (9.3.9[2], last page) and change the element sizes as shown below. →

Element Size (mm)	Number of Nodes	Tip Deflection (mm)
5	727	0.74767
4	1303	0.74914
3.3	2803	0.75001
3	3805	0.75018
2.4	5827	0.75062
2	9530	0.75082
1.8	12779	0.75100
1.6	18902	0.75115
1.4	27573	0.75128



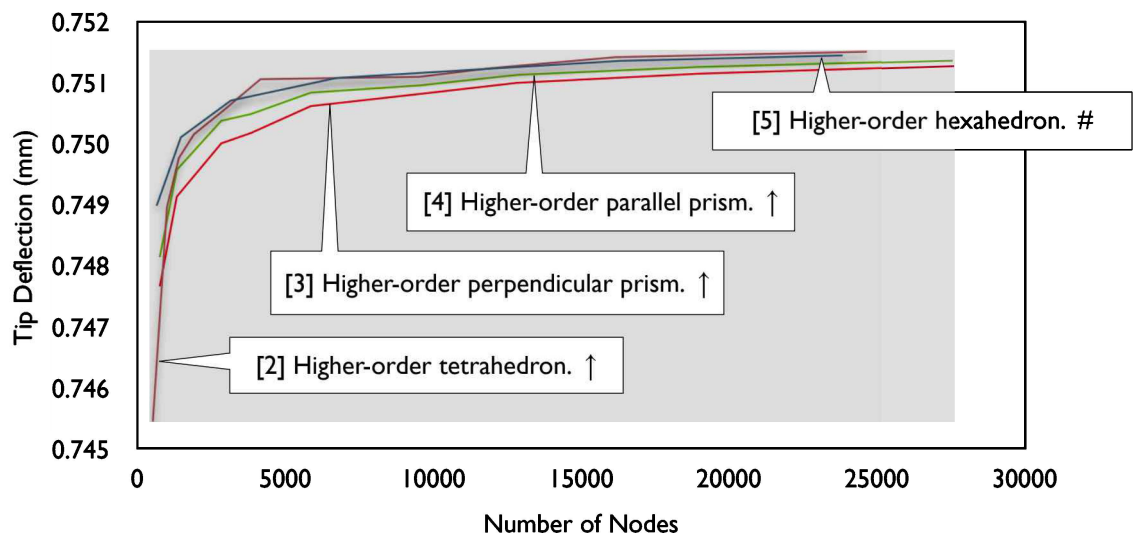
9.3.13 Comparison: Lower-Order Elements

[1] The chart below is made by a collection of the convergence curves in 9.3.5 to 9.3.8 (pages 355-358) to compare the convergence behaviors of the lower-order elements. The order of the convergence speed is, from fast to slow, hexahedron, parallel prism, perpendicular prism, and tetrahedron. The differences among them are obvious. The lower-order tetrahedron converges so poorly that it is not practically useful. As a guideline, NEVER use lower-order tetrahedral elements. ↓



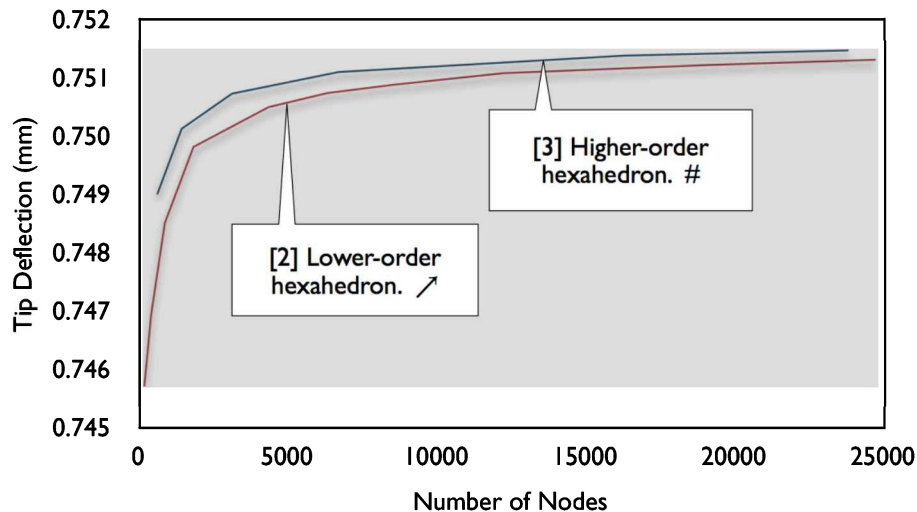
9.3.14 Comparison: Higher-Order Elements

[1] The chart below is made from a collection of the convergence curves in 9.3.9 to 9.3.12 (pages 359-360) to compare the convergence behaviors of the higher-order elements. The differences among them are not obvious but still distinguishable. In contrast to the lower-order tetrahedron, the higher-order tetrahedron is still practically useful as long as the mesh is fine enough. ↓



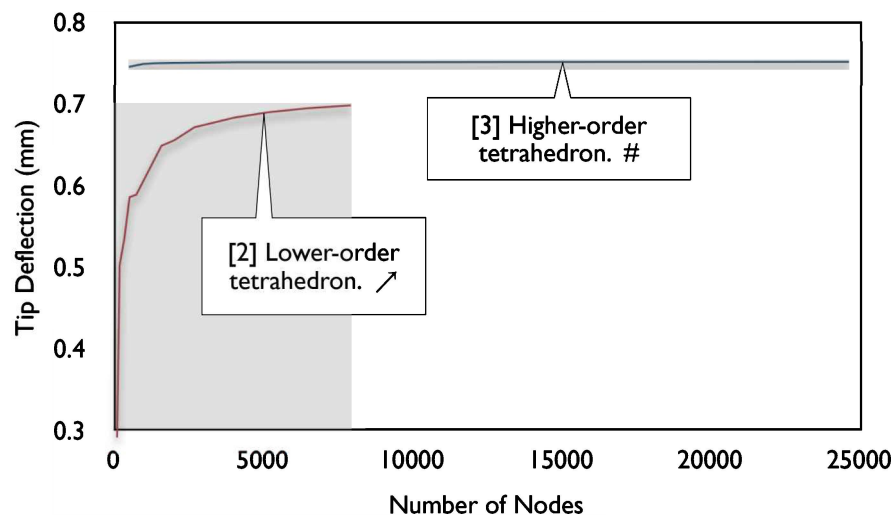
9.3.15 Comparison: Hexahedra

[1] The chart below is made by a collection of the convergence curves in 9.3.5 (page 355) and 9.3.9 (page 359) to compare the convergence behaviors between the lower- and higher-order hexahedra. It is obvious that the higher-order hexahedral element is better than the lower-order hexahedral, but the difference is not so dramatic as the tetrahedral element (see 9.3.16, this page). ↓



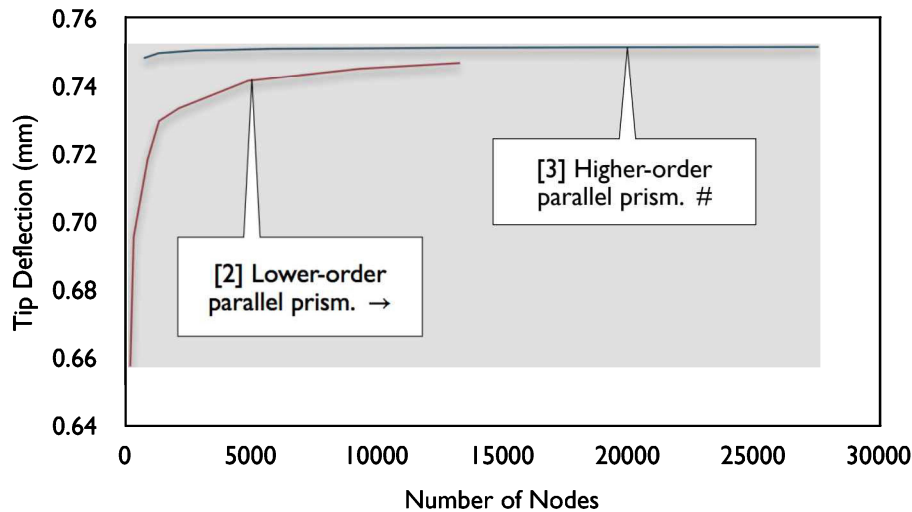
9.3.16 Comparison: Tetrahedra

[1] The chart below is made from a collection of the convergence curves in 9.3.6 (page 356) and 9.3.10 (page 359) to compare the convergence behaviors between the lower- and higher-order tetrahedra. It is obvious that higher-order element is much better than the lower-order one. Remember: NEVER use lower-order tetrahedral elements. ↓



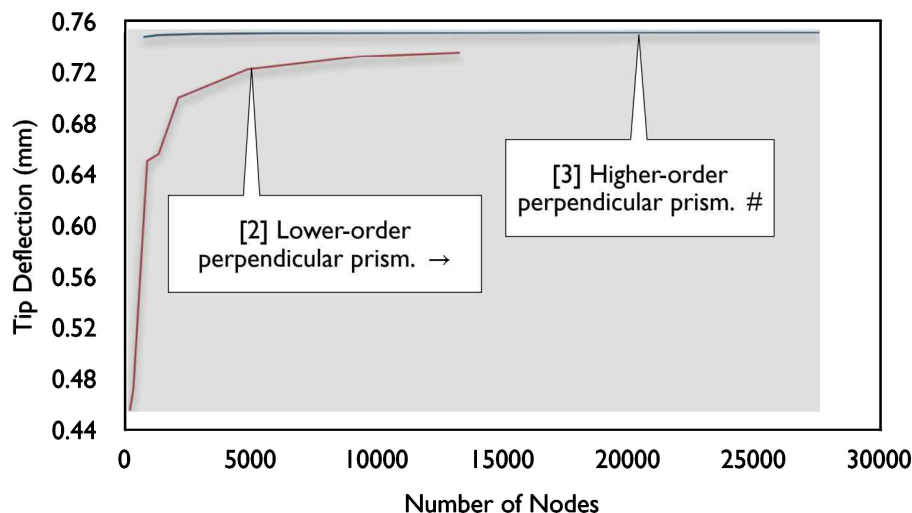
9.3.17 Comparison: Parallel Prisms

[1] The chart below is made by a collection of the convergence curves in 9.3.7 (page 357) and 9.3.11 (page 360) to compare the convergence behaviors between the lower- and higher-order parallel prisms. It is obvious that higher-order element is much better than the lower-order one. Like lower-order tetrahedral, lower-order prismatic elements are not recommended. ↓



9.3.18 Comparison: Perpendicular Prisms

[1] The chart below is made by a collection of the convergence curves in 9.3.8 (page 358) and 9.3.12 (page 360) to compare the convergence behaviors between the lower- and higher-order perpendicular prisms. It is obvious that the higher-order element is much better than the lower-order one. Like lower-order tetrahedral, lower-order prismatic elements are not recommended. ↓



9.3.19 Summary and Guidelines

Summary and Guidelines

[1] Combining the observations in Section 3.5 and this section, we may summarize the conclusions as follows: (a) Never use lower-order tetrahedra or triangles. (b) Higher-order tetrahedra or triangles are as good as other elements as long as the mesh is fine enough. In cases of coarse mesh, however, they perform poorly and are not recommended. (c) Lower-order prisms are not recommended. (d) Lower-order hexahedra and quadrilaterals can be used, but they are not as efficient as their higher-order counterparts. (e) Higher-order hexahedra, parallel-prisms, and quadrilaterals are among the most efficient elements we have discussed so far. Mesh your models with these elements whenever possible. If that is not possible, then at least try to achieve a higher-order hexahedra-dominant or quadrilateral-dominant mesh. ↓

Remark: CPU Time

[2] In Section 3.5 and this section, comparisons among elements are made under the same number of nodes. More reasonable comparisons should be made under the same CPU time. For a simulation task, the CPU time consists of three parts. First, the time required to establish Eq. 1.3.1(1) (page 35). It may involve numerical integrations for each element. This part of CPU time depends on the total number of elements as well as the number of integration points of each element. Second, the time required to solve the equation. This part of CPU time is determined solely by the number of degrees of freedom, which is in turn determined by the number of nodes and the dimensionality (2D or 3D). Third, the others (housekeeping, overhead, etc).

For small problems, the overall CPU time is dominated by the third part. That is why we didn't use CPU time for comparison, since all cases are small when coarsely meshed. For large problems, the third part is negligible and the CPU time is essentially the sum of the first two parts.

Therefore, strictly speaking, our comparison was not perfectly accurate. Nevertheless, the discussions and conclusions in this section pretty much reflect the reality. These guidelines should be useful. ↓

Wrap Up

[3] Save the project and exit Workbench. #

Section 9.4

Review

9.4.1 Keywords

Choose a letter for each keyword, from the list of descriptions

- | | |
|---|-----------------------------------|
| 1. () Convergence Criteria | 6. () Patch Conforming Method |
| 2. () Displacement Convergence Criterion | 7. () Patch Independent Method |
| 3. () Force Convergence Criterion | 8. () Perpendicular Prisms |
| 4. () Hex Dominant Method | 9. () Skewness |
| 5. () Parallel Prisms | 10. () Sweep Thin Method |

Answers:

1. (B) 2. (C) 3. (D) 4. (H) 5. (I) 6. (F) 7. (G) 8. (J) 9. (A) 10. (E)

List of Descriptions

(A) A measure of mesh quality, calculated for each element according to its geometry. Its value ranges from 0 to 1, the smaller the better. Elements of skewness of more than 0.95 are considered unacceptable.

(B) In nonlinear simulation, the loading is divided into substeps and applied substep by substep. By default, a substep is said to be complete when both displacement convergence criterion and force convergence criterion are met during the iterations.

(C) During the iterations of a substep of a nonlinear simulation, the displacement convergence criterion is met when the increment of displacement is less than a criterion, which is, by default, 0.5% of maximum displacement.

(D) During the iterations of a substep of a nonlinear simulation, the force convergence criterion is met when the unbalanced force is less than a criterion, which is, by default, 0.5% of applied force.

(E) **Sweep** mesh control method can be classified into **Sweep** and **Sweep Thin**. **Sweep** allows a more complex sweeping path while **Sweep Thin** allows only a simple sweeping path. The advantage of **Sweep Thin** is that it allows multiple faces as source or target while **Sweep** allows only one face for both source and target.

(F) A mesh control method. It meshes all the faces of the body with triangles; the triangles then "grow" inward to create tetrahedra. In this way, the shapes of the faces are respected (preserved).

(G) A mesh control method. It creates tetrahedra from inside out. The outermost nodes are then projected onto the boundary faces and the element edges are created. In this way, the mesh's outline may be different from the original geometry.

(H) A mesh control method. It meshes a body with Patch Conforming method first and then combines tetrahedra to form hexahedra. It usually leaves some tetrahedra that cannot be combined to form hexahedra.

(I) When a body is meshed with prismatic elements and the prisms are oriented such that their heights are parallel to the bending direction, the prismatic elements oriented in this way are referred to as parallel prisms. (Note: this term is used only in this book.)

(J) When a body is meshed with prismatic elements and the prisms are oriented such that their heights are perpendicular to the bending direction, the prismatic elements oriented in this way are referred to as perpendicular prisms. (Note: this term is used only in this book.)

9.4.2 Additional Workbench Exercises

Convergence Study for Higher-Order 2D Elements

In Section 9.3, we study the convergence of 3D elements, both higher-order and lower-order elements. In Section 3.5, we study the convergence of 2D elements only for the lower-order elements. We haven't studied the higher-order 2D elements yet. Conduct a study of the higher-order 2D elements.

Chapter 10

Buckling and Stress Stiffening

Functionality, safety, and reliability are the main purposes of structural simulations. Stresses usually relate to safety and reliability. In the 3D truss example (Section 7.2), calculated stresses are well below the material's yield strength (7.2.12[4], page 292). Can we conclude that the design is safe? Not yet. For any structural members (particularly slender or thin members) subject to compressive stresses, we need to check their stability before concluding their safety. This chapter mainly discusses *stability analysis*, or *buckling analysis*.

Buckling can be viewed as an ultimate case of a more general effect, called *stress stiffening*: a structure member's bending stiffness increases with increasing axial tensile stress, and, on the other hand, the member's bending stiffness decreases with the increasing compressive stress. Buckling occurs when the compressive stress reaches a level such that the bending stiffness reduces to zero; in that situation, the applying load is called a *buckling load* and the corresponding deforming shape is called a *buckling mode*. The purpose of buckling analyses is to find the buckling loads and the corresponding *buckling mode*.

Purpose of This Chapter

The main purpose of this chapter is to introduce *linear buckling analysis* (also known as *eigenvalue buckling analysis*). Since buckling can be viewed as an ultimate case of stress stiffening, the discussion will start with a thorough understanding of stress stiffening. As usual, the concepts are introduced using step-by-step exercises.

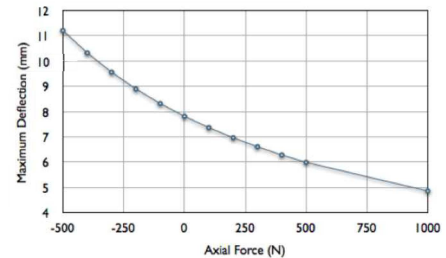
About Each Section

Section 10.1 introduces the stress stiffening effects, using a simply supported beam as an example. The results of these nonlinear analyses can be used to predict the buckling load using an extrapolation method; this procedure is called a *nonlinear buckling analysis*. A linear buckling analysis is then carried out to find the buckling loads and buckling modes. Compared with nonlinear buckling analyses, linear buckling analysis tends to overestimate the buckling load.

Section 10.2 performs a linear buckling analysis on the 3D truss structure introduced in Section 7.2. Section 10.3 carries out a linear buckling analysis on the beam bracket which has been discussed in Sections 4.1, 5.1, and 6.2.

Section 10.1

Stress Stiffening



This section introduces *stress stiffening effect*, which is closely related to buckling. Stress stiffening effect is often observed in slender or thin structural members, such as cables, shells, columns, walls, towers, trusses, etc. In a slender structural member, such as a column, the bending stiffness is affected by its axial stress. In a thin structure member, such as a wall, the bending stiffness is affected by its in-plane stress. More specifically, when subject to axial (or in-plane) tension, the bending stiffness tends to increase, while when subject to axial (or in-plane) compression, the bending stiffness tends to decrease.

In this section, we will use a simply supported slender beam to demonstrate the stress stiffening effects.

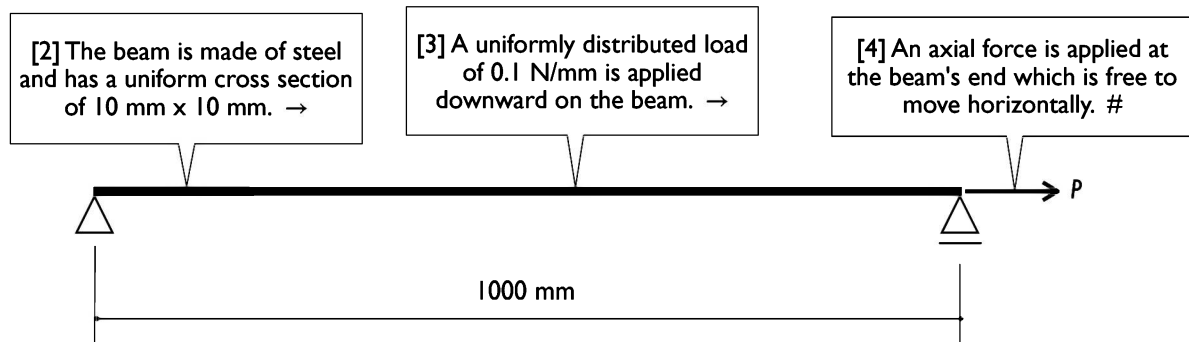
10.1.1 About the Simply Supported Beam

[1] Consider a simply supported beam shown below [2-4]. The beam is made of steel and has a uniform cross section of 10 mm x 10 mm [2]. A uniformly distributed load of 0.1 N/mm is applied downward on the beam [3]. An axial force is applied at the beam's end which is free to move horizontally [4].

The vertical load, which causes bending, is a constant 0.1 N/mm, while the horizontal force P will change from -500 N to 1000 N. Note that the negative P produces a compressive axial stress and the positive P produces a tensile axial stress. We will examine the maximum vertical deflection occurring at the middle of the span.

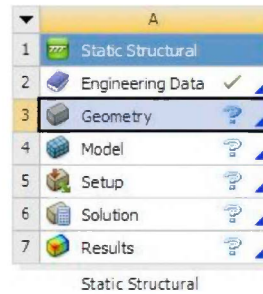
Let δ_0 be the beam's middle-span deflection when $P = 0$. Then, when the beam is subject to a positive P , we will obtain a deflection less than δ_0 . We can then conclude that the bending stiffness increases with the increasing tensile axial stress. On the other hand, when the beam is subject to a negative (compressive) P , we will obtain a deflection larger than δ_0 . We can then conclude that the bending stiffness decreases with the increasing compressive axial stress. This effect is called the *stress stiffening*.

Since the bending stiffness decreases with the increasing compressive axial stress, you may raise a question: How large the compressive force P will cause the bending stiffness to reduce to zero? A zero bending stiffness implies an unstable structure, because a small lateral load would cause an infinitely large deflection. This phenomenon is called the *buckling*. ↓

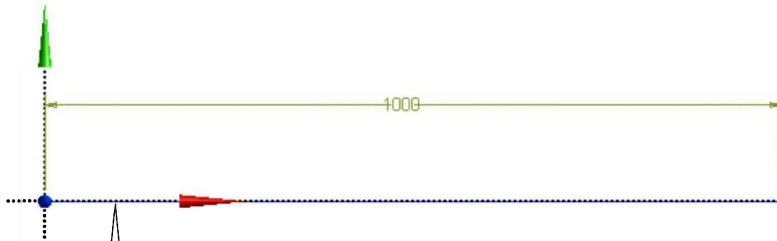


10.1.2 Start a New Project

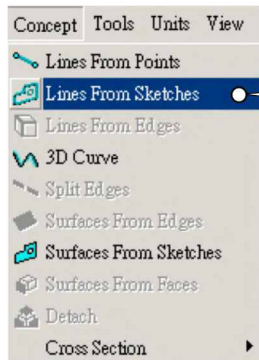
[1] Launch Workbench. Create a **Static Structural** system. Save the project as **SimpleBeam**. Start up DesignModeler. #



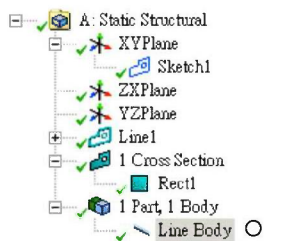
10.1.3 Create a Line in DesignModeler



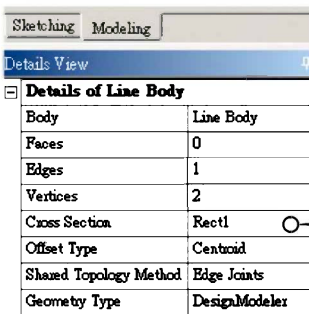
[1] Select **Millimeter** as the length unit. On **XYPlane**, draw a straight line of length 1000 mm along X-axis. →



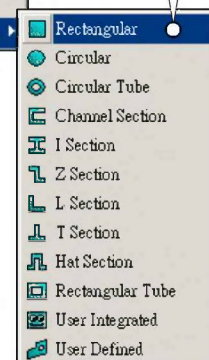
[2] Select **Concept/Lines From Sketches** and create a line body using the newly created sketch (**Sketch1**). Remember to click **Generate**. ↓



[3] Select **Concept/Cross Section/Rectangular** and create a rectangular cross section of 10 mm x 10 mm (default). ✓



[4] With **Line Body** in the project tree highlighted, assign the newly created cross section to the line body. Close DesignModeler. #



10.1.4 Set Up Supports

[1] Start up **Mechanical** and select **mm-kg-N-s** unit system. ✓

Details of "Simply Supported"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Simply Supported
Suppressed	No

[2] With **Static Structural** in the project tree highlighted, insert a **Supports/Simply Supported**. Apply this condition at the beam's left end. You may need to use the vertex selection filter. A **Simply Supported** vertex prohibits translations in all directions, but allows rotations in all directions. →

Details of "Displacement"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
Y Component	0. mm (gamped)
Z Component	0. mm (gamped)
Suppressed	No

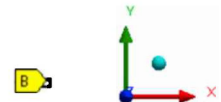
[3] Insert a **Displacement**. Apply this condition at the beam's right end. Type zeros (i.e., fixed) for the Y- and Z-displacement and leave **Free** for the X-displacement. →

Details of "Fixed Rotation"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Fixed Rotation
Coordinate System	Global Coordinate System
Rotation X	Fixed
Rotation Y	Free
Rotation Z	Free
Suppressed	No

[4] Insert a **Supports/Fixed Rotation**. Apply this condition at either end of the beam. Here, we apply at the left end. Select **Free** for the Y- and Z-rotation and leave **Fixed** for the X-rotation. ↓

A: Static Structural
Static Structural
Time: 1. s

A Fixed Rotation: 0. °
B Displacement
C Simply Supported: 0. mm



Provide Enough Supports to Avoid Rigid Body Modes

[5] In 3.1.8[7] (page 116), we mentioned that it is a good practice to provide enough supports. This becomes a necessity when working on buckling or modal analyses, where rigid body modes are not automatically eliminated with weak springs.

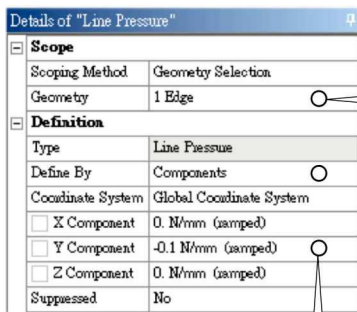
In this case, newcomers often fail to fix Z-displacements in step [3] and the X-rotation in step [4]. Without these supports, a nonlinear simulation would run into convergence difficulties. In case of buckling or modal analyses, rigid body modes would present.

What's wrong with the presence of rigid body modes? The answer depends on the type of simulation. For **Static Structural** simulations, Workbench can add weak springs to prevent an uncontrolled large amount of rigid body motions (3.1.8[7], page 116) and allow a small amount of rigid body motion (3.1.10[2], page 117).

For **Eigenvalue Buckling** and **Modal** simulations, rigid body modes are trivial and appear to be harmless. The buckling load corresponding to a rigid body mode is zero, and the natural frequency corresponding to a rigid body mode is also zero. However, presence of rigid body modes may deteriorate the numerical accuracy.

For **Transient Structural** simulations, we usually don't need to artificially eliminate rigid body modes if they exist naturally; let the rigid body modes be present, and the program will take care of them nicely. The exercises in Sections 12.4, 15.2, and 15.3 provide examples for these situations. #

10.1.5 Set Up Loads

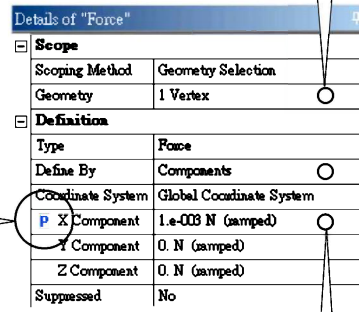


[1] Apply a **Loads/Line Pressure** on the beam. ↓

[5] Click to set **X Component** as an input parameter. #

[2] Type -0.1 (N/mm) for **Y Component**. ↗

A: Static Structural
Static Structural
Time: 1 s



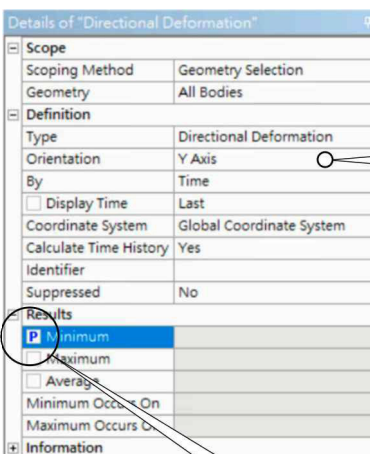
[3] Apply a **Force** at the beam's right end. ↓

[4] Type a small value, here 0.001 (N), for **X Component**. Workbench doesn't allow all-zero components, therefore we type a small value instead. ↖

- A** Simply Supported: 0. mm
- B** Displacement
- C** Fixed Rotation: 0. °
- D** Line Pressure: 0.1 N/mm
- E** Force: 1.e-003 N

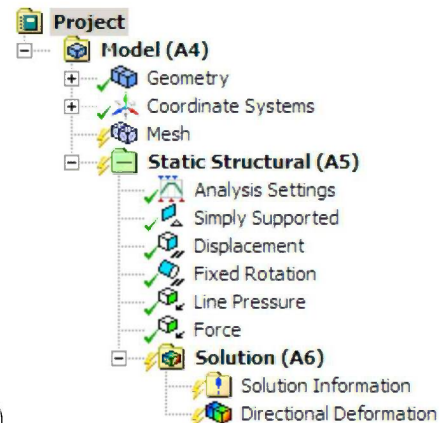


10.1.6 Set Up Solution Objects

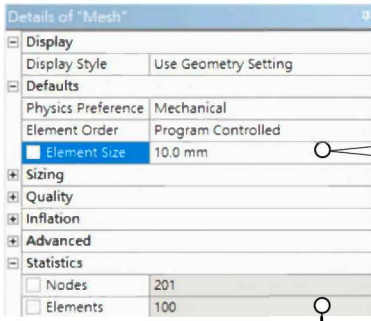


[1] With **Solution** highlighted, insert a **Deformation/Directional**. Set **Orientation** to **Y Axis**. ↓

[2] Click to set **Minimum** as an output parameter. The Y-deformation has a negative value, so we look at the **Minimum** value. #

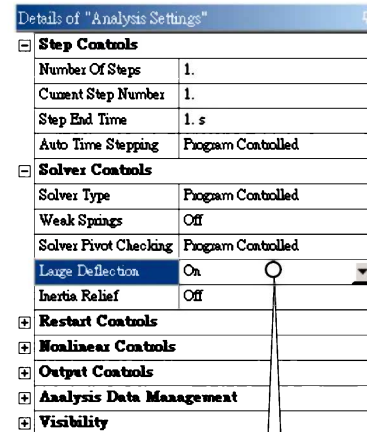


10.1.7 Set Up Mesh and Turn on Large Deflection Effect



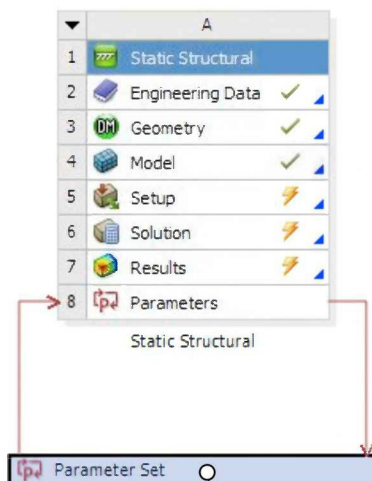
[1] With **Mesh** in the project tree highlighted, set element size to 10 (mm). Generate mesh. ↓

[2] It generates 100 beam elements. We will use this mesh for the simulations in this section. →



[3] With **Analysis Settings** in the project tree highlighted, turn on **Large Deflection**. This turns on geometrical nonlinearity. The stress stiffening effect is due to force-geometry interactions; **Large Deflection** must be turned on to include the stress stiffening effects. Close **Mechanical**. #

10.1.8 Create Design Points



[1] In the project schematic, double-click **Parameter Set**. ↗

	A	B	C
1	Name	P1 - Force X Component	P2 - Directional Deformation Minimum
2	Units	N	mm
3	DP 0 (Current)	0.001	
4	DP 1	100	
5	DP 2	200	
6	DP 3	300	
7	DP 4	400	
8	DP 5	500	
9	DP 6	1000	
10	DP 7	-100	
11	DP 8	-200	
12	DP 9	-300	
13	DP 10	-400	
14	DP 11	-500	
*			

[2] Select **tonne-mm-s** as the project unit. In **Table of Design Points**, add 11 design points by typing these values. ↓

⚡ Update All Design Points

[3] Click **Update All Design Points**. It takes a while. #

10.1.9 The Results and Discussion

Table of Design Points			
	A	B	C
1	Name	P1 - Force X Component	P2 - Directional Deformation Minimum
2	Units	N	mm
3	DP 0 (Current)	0.001	-7.8114
4	DP 1	100	-7.3626
5	DP 2	200	-6.9624
6	DP 3	300	-6.6034
7	DP 4	400	-6.2796
8	DP 5	500	-5.9863
9	DP 6	1000	-4.8551
10	DP 7	-100	-8.3184
11	DP 8	-200	-8.8953
12	DP 9	-300	-9.5571
13	DP 10	-400	-10.321
14	DP 11	-500	-11.205
15			

[2] And the deflections are used as the vertical axis. ↓

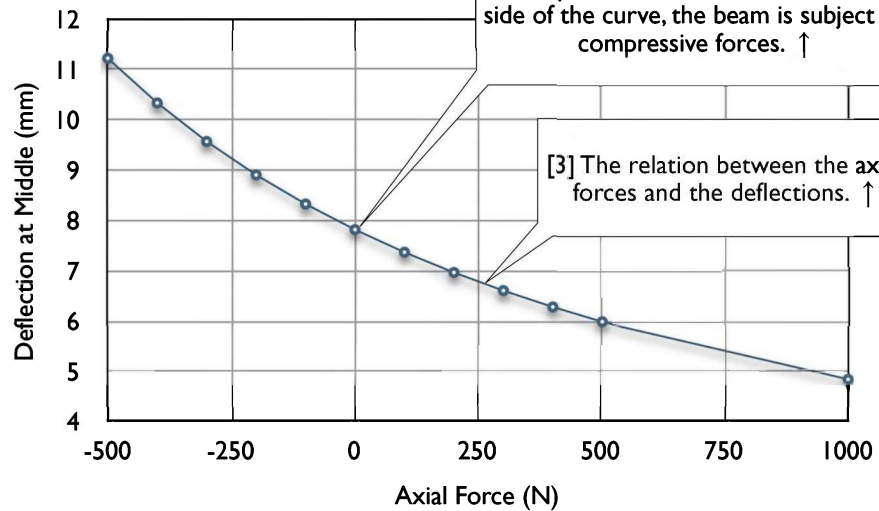
Parameters Studies with Workbench

[5] We've demonstrated a useful feature of Workbench: parameters study, mentioned in 8.1.5[12-14], page 317.

To study the relations among parameters, you may create various "design points" and then issue an **Update All Design Points** command. Workbench will do the rest of the work.

The resulting data can be copied and pasted to a spread sheet application, such as **Excel**, for plotting a chart. ↓

[1] We now plot a chart as shown in [3]. Note that, in [3] the forces are used as the horizontal axis... ↗



[4] This is the point of zero axial force. To the right side of the curve, the beam is subject to tensile forces; to the left side of the curve, the beam is subject to compressive forces. ↑

[3] The relation between the axial forces and the deflections. ↑

Stress Stiffening Effects

[6] The curve [3] manifests the stress stiffening effects. As the tensile axial force increases, the deflection decreases, indicating an increase of bending stiffness. On the other hand, as the compressive axial force increases, the deflection increases, indicating a decrease of bending stiffness. When the compressive axial force reaches a certain point, the bending stiffness decreases so much that the deflection is enlarged dramatically. You may raise a question: what is the compressive force such that the bending stiffness completely vanishes? A zero bending stiffness implies an unstable structure: a small vertical load would cause the beam to collapse. This phenomenon is called the *buckling* and the compressive force causing the structure to buckle is called the *buckling load*, or *critical load*. ↩

[7] We can continue the above process and extend the curve ([3], last page) leftward until a vertical asymptote can be drawn. The force value intercepted by the asymptote will be the buckling force. And this procedure is basically a *nonlinear buckling analysis*^[Ref 1].

Workbench provides an **Eigenvalue Buckling** system to assess the linear buckling load. A linear buckling analysis usually takes much less computing time than a nonlinear buckling analysis but usually overestimates the buckling load.

The linear buckling theory predicts the buckling load as

$$P_{\text{buckling}} = \frac{\pi^2 EI}{L^2} = \frac{\pi^2 (200,000)(833.33)}{(1000)^2} = 1645 \text{ N} \quad (1)$$

Note that the above calculation doesn't include the distributed beam load (0.1 N/mm). Although tending to overestimate the buckling load, the linear buckling analysis is useful for two reasons: (a) It is computationally much cheaper than a nonlinear buckling analysis and should be run as a first step to estimate the buckling load. (b) It can be used to determine the possible buckling mode.

We will proceed to demonstrate the linear buckling analysis for this case in the rest of the section. #

10.1.10 Set Up Project Schematic

[1] Click **Project** to return to **Project Schematic**. ←

[2] Create a new **Static Structural** system by double-clicking it in **Toolbox**. ←

[3] Drag-and-drop the **Geometry** cell from the old system to the **Geometry** cell of the new system. The two systems now share the same geometry. ↓

[4] Right-click **Solution** and select **Transfer Data To New/Eigenvalue Buckling**. ←

The screenshot shows the Project Schematic with two systems, A and B. System A contains cells: Static Structural, Engineering Data, Geometry, Model, Setup, Solution, Results, and Parameters. System B contains: Static Structural, Engineering Data, Geometry, Model, Setup, Solution, and Results. A red arrow indicates the transfer of the Geometry cell from System A to System B. A right-click context menu is open on the Solution cell of System B, with the 'Transfer Data To New' option selected, leading to a list of analysis types where 'Eigenvalue Buckling' is highlighted. A 'Project' button is shown in the top right corner.

[6] Double-click **Model** (or any cell below **Model**) to start up **Mechanical**. #

[5] An **Eigenvalue Buckling** system is created and the **Solution** of the **Static Structural** system is linked to the **Setup** of the **Eigenvalue Buckling** system (see 10.1.14[3], next page). ←

10.1.11 Set Up Supports

[1] Highlight **Static Structural** and repeat the steps 10.1.4[2-4] (page 370). #

B: Static Structural
Static Structural
Time: 1. s

Fixed Rotation: 0.°
Simply Supported: 0. mm
Displacement

10.1.12 Set Up Load

[1] With **Static Structural** highlighted, Insert a **Force**. Select the beam's right end. →

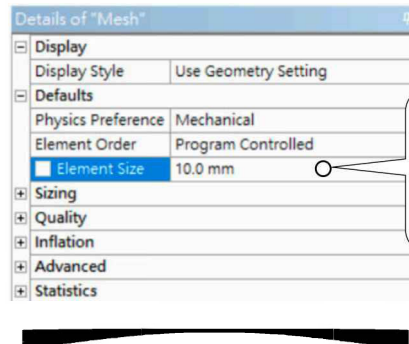
[2] Type -100 (N) for **X Component**. This is an arbitrary compressive value. After a buckling analysis, the buckling load will be reported as a multiplier of this applied load. #

Details of "Force"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	-100. N (ramped)
Y Component	0. N (ramped)
Z Component	0. N (ramped)
Suppressed	No

B: Static Structural
Force
Time: 1. s

Force: 100. N
Components: -100., 0., 0. N

10.1.13 Set Up Mesh



[1] With **Mesh** in the project tree highlighted, set element size to 10 (mm). #

10.1.14 Specify Number of Buckling Modes and Solve

Project

- Model (B4, C4)
 - Geometry
 - Coordinate Systems
 - Mesh
 - Static Structural (B5)
 - Analysis Settings
 - Simply Supported
 - Displacement
 - Fixed Rotation
 - Force
 - Solution (B6)
 - Solution Information
 - Eigenvalue Buckling (C5)
 - Pre-Stress (Static Structural)
 - Analysis Settings
 - Solution (C6)
 - Solution Information

Details of "Analysis Settings"

Options	
Max Modes to Find	4
Solver Controls	
Solver Type	Program Controlled
Include Negative Load Multiplier	Program Controlled
Output Controls	
Analysis Data Management	

[3] The name of this object indicates that the stress results of **Static Structural** are used as a pre-stress for **Eigenvalue Buckling** (see 10.1.10[5], last page). ↗

[4] Highlight **Eigenvalue Buckling** and click **Solve**. This also triggers the solving of **Static Structural** (for calculating the pre-stress). If you highlight **Static Structural** when you click **Solve**, it would solve the **Static Structural** only. ↓

[1] Highlight **Analysis Settings** of **Eigenvalue Buckling**. ↓

[2] Type 4 for **Max Modes to Find**. ↑

Number of Buckling Modes

[5] In this case, the first buckling mode (mode with the smallest buckling load) is what we are interested in. Other modes have little practical usage in this case.

Due to the same dimensions of the cross section in Y- and Z-direction, buckling modes appear as pairs, that is, the first two modes are identical, the third and fourth modes are identical, and so on.

The reason we want to evaluate the third and the fourth modes is to make sure they are far beyond our concern. #

10.I.15 View the Results

[1] Highlight **Solution**.

[2] These are the **Load Multipliers** for the first four buckling modes; i.e., a buckling load is equal to the applied load times a **Load Multiplier**. For example, the first buckling load is 1644.5 N (100×16.445), which is consistent with the result in Eq. 10.1.9(1), page 374.

[3] Click **Mode** to select all modes and right-click-select **Create Mode Shape Results**.

[4] Click **Solve**.

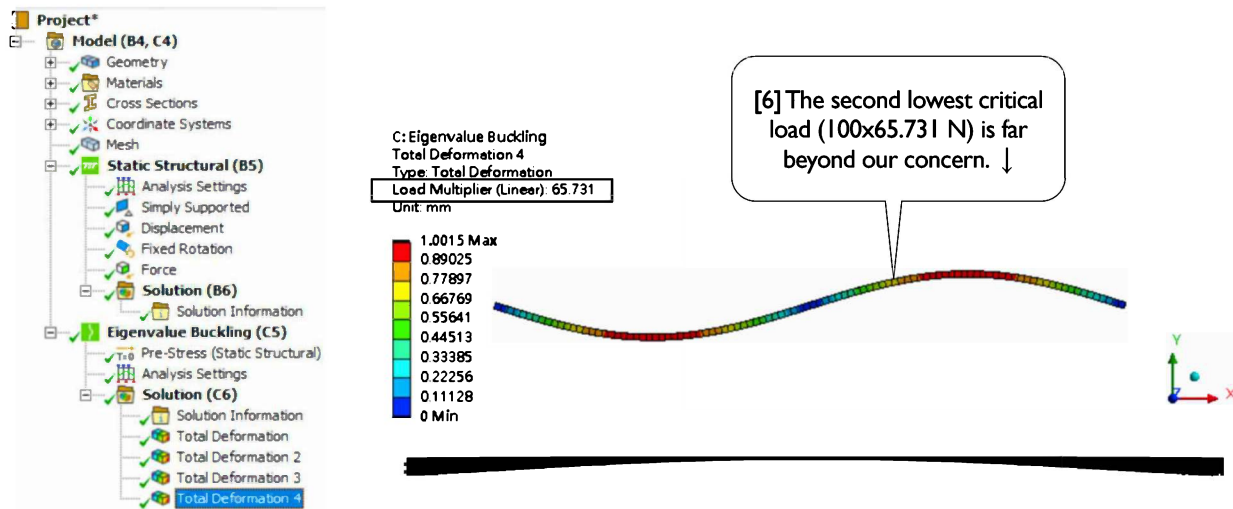
Mode	Load Multiplier
1	16.445
2	16.445
3	65.731
4	65.731

[5] Now, you can click a result object to view a buckling mode.

C: Eigenvalue Buckling
Total Deformation
Type: Total Deformation
Load Multiplier (Linear): 16.445
Unit: mm

1.3309 Max
1.183
1.0351
0.88725
0.73938
0.5915
0.44363
0.29575
0.14788
0 Min

Mode	Load Multiplier
1	16.445
2	16.445
3	65.731
4	65.731



Buckling Mode Shapes

[7] When displaying the buckling mode shapes, Workbench scales the values of deformation such that the maximum deformation is approximately 1.0. The values of deformation have no physical significance. It is the mode shapes that are useful. Similarly, the stresses or strains calculated in linear buckling analyses have no physical meaning. ↓

Wrap Up

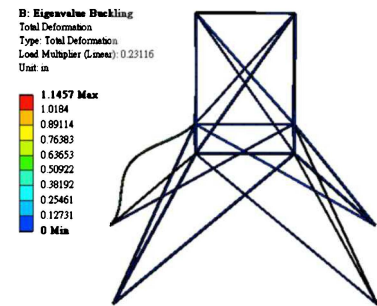
[8] Save the project and exit Workbench. #

Reference

1. All Help>Mechanical APDL>Structural Analysis Guide>7.2. Performing a Nonlinear Buckling Analysis

Section 10.2

3D Truss



10.2.1 About the 3D Truss

[1] In Section 7.2, we analyzed a 3D truss; the stresses range from -25,415 psi to +13,962 psi (7.2.12[4], page 292). The structure seems safe, since the structural steel's yield strength can be as high as 40,000 psi for both tension and compression. However, stress is only one design consideration. Structural stability must be verified whenever compressive members are involved in the structure system.

Let's make some simple calculations to check whether structural stability should be an issue, by considering the member that has the maximum compressive force, which is $P = 15,802$ lb (see [2]). →

A: Static Structural

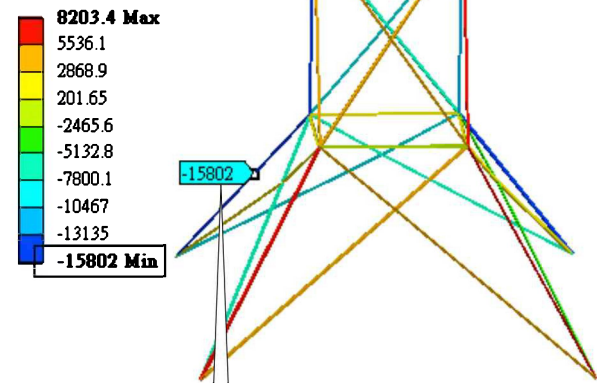
Axial Force

Type: Directional Axial Force (Unaveraged) (X Axis)

Unit: lbf

Solution Coordinate System

Time: 1



[2] Maximum compressive force is 15,802 lbf. This picture is the same as 7.2.13[6], page 294. ↓

[3] The member (of maximum compressive force [2]) has a length of 133.46 inches and a cross section ($L \ 1\frac{1}{2} \times 1\frac{1}{2} \times \frac{1}{4}$) of an area moment of inertia of 0.13852 in⁴. The structural steel's Young's modulus is 29,000,000 psi. Its buckling load, according to a linear buckling theory, is estimated to be

$$P_{\text{buckling}} = \frac{\pi^2 EI}{L^2} = \frac{\pi^2 (29,000,000)(0.13852)}{(133.46)^2} = 2,226 \text{ lb} = 0.14(15,802) = 0.14P$$

In other words, merely 14% of the design loads (see 7.2.1[1], page 283) would cause at least one of the structural members to buckle. It is now obvious that the structural stability might be a problem.

In this section, we want to perform a linear buckling analysis for the entire 3D truss structure, rather than a single member. The linear buckling analysis will result in a higher multiplier than 14% (10.2.4[6], page 382). Two factors cause the deviation from the above simple calculation. First, the rigid joints provide additional rigidity, increasing the buckling load. Second, even assuming pin-jointed, this truss is a statically indeterminate structure; buckling of one member does not necessarily cause a buckling of the entire structure.

The unit system used in this section is **in-lbm-lbf-s. #**

10.2.2 Resume the Project **Truss**

[1] Launch Workbench, open the project **Truss** (which was saved in Section 7.2), and set up an **Eigenvalue Buckling** system shown in [2-5]. ✓

[2] From **Toolbox**, drag **Eigenvalue Buckling** and drop here. ✓

[3] An **Eigenvalue Buckling** system is created. ↓

[4] This link means that the stress results (of **Static Structural**) are used as a pre-stress of **Eigenvalue Buckling**. ↑

[5] Double-click **Setup** (or any cells below) to start up **Mechanical**. Select the **in-lbm-lbf-s** unit system. #

10.2.3 Perform Buckling Analysis

[1] With **Mesh** in the project tree highlighted, set element size to 10 (in) and generate the mesh. The reason we use smaller element size (instead of one element for each member) is to display the deformation in a visually more realistic fashion. ✓

[2] Highlight **Analysis Settings** of **Eigenvalue Buckling**. ↓

[3] Type 6 for **Max Modes to Find**. ↓

[4] Select **No** for **Include Negative Load Multiplier**. We're not interested in negative load multipliers. ←

[5] Highlight **Eigenvalue Buckling** and click **Solve**. #

10.2.4 View the Results

[4] Click **Solve**.

[3] Click **Mode** to select all modes and right-click-select **Create Mode Shape Results**.

[2] The multipliers for the first six buckling modes.

[1] Highlight **Solution**.

Graph: Buckling Multiplier

Mode	Load Multiplier
1	0.23116
2	0.25247
3	0.37218
4	0.38788
5	0.4001
6	0.46573

[5] Click **Total Deformation** to view the first buckling mode.

Graph: Total Deformation

Mode	Load Multiplier
1	0.23116
2	0.25247
3	0.37218
4	0.38788
5	0.4001
6	0.46573

[6] Buckling will occur when 23% of design load is applied on the structure. The **Load Multiplier** can be viewed as a safety factor as far as the buckling is concerned. The structure is not safe. ↓

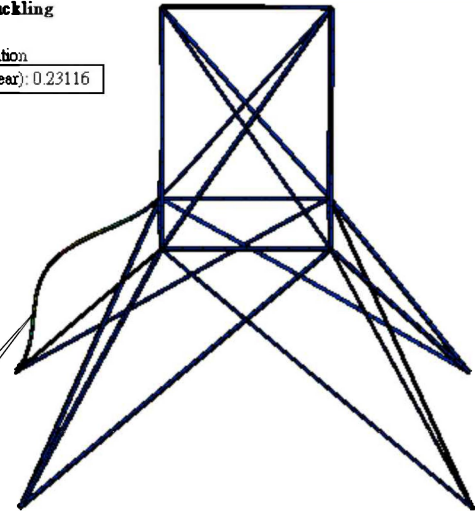
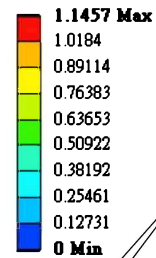
B: Eigenvalue Buckling

Total Deformation

Type: Total Deformation

Load Multiplier (Linear): 0.23116

Unit: in



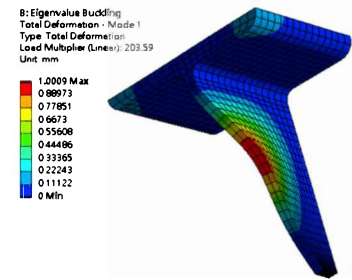
[7] The first buckling mode. ↓

Wrap Up

[8] Save the project and exit Workbench. #

Section 10.3

Beam Bracket



10.3.1 About the Beam Bracket

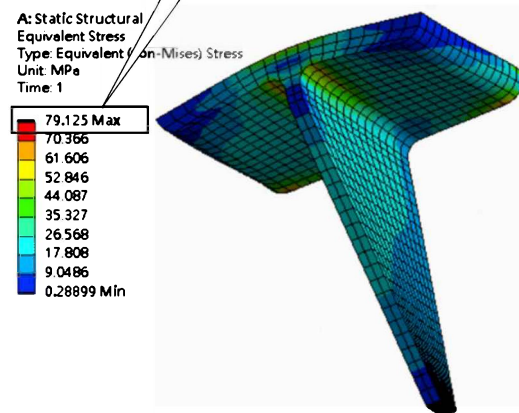
[1] In Section 5.1, we simulated a beam bracket using a 3D solid model, and in Section 6.2 using a surface model. The maximum von Mises stress is 83 MPa [2], well below the yield strength, 250 MPa.

By examining **Minimum Principal Stress** [3], we see that the web is subject to compressive stress; its magnitude is about 24 MPa [4]. It is a good practice that an engineer always checks the structural stability whenever compressive stresses exist, unless he has enough experience to judge that the stability is not an issue and therefore the checking is not necessary.

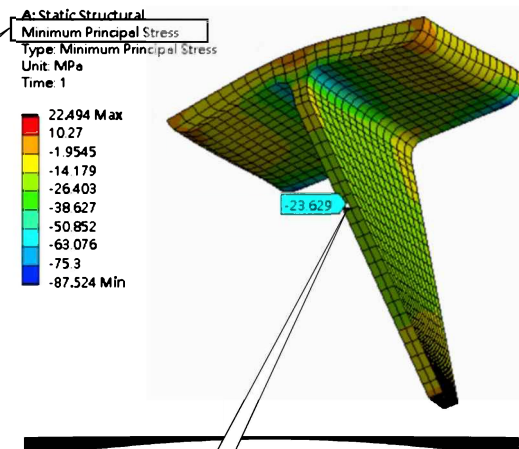
In this section, we want to make sure that, under the design load, the web does not buckle.

The unit system **mm-kg-N-s** is used in this section. ↗

[2] The maximum von Mises stress. This picture is a duplicate of 5.1.13[9], page 218. ✓



[3] **Minimum Principal Stress** is used to examine compressive stresses. ↓

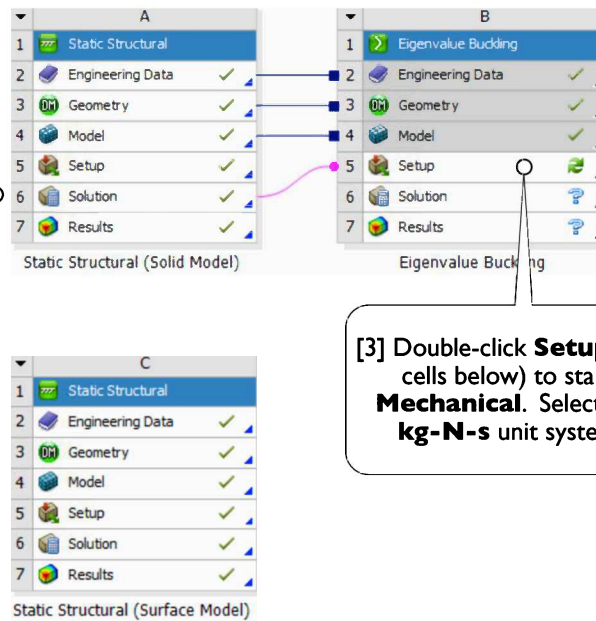


[4] Compressive stress at the web. #

10.3.2 Resume the Project **Bracket**

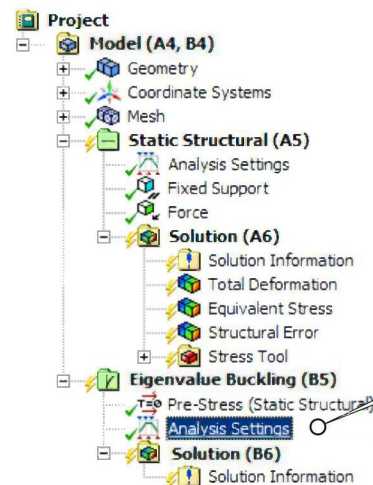
[1] Launch Workbench. Open the project **Bracket**, which was saved in Section 6.2. ↓

[2] From **Toolbox**, drag **Eigenvalue Buckling** and drop here. →



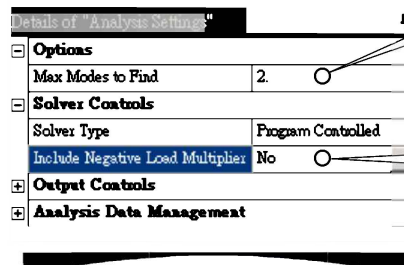
[3] Double-click **Setup** (or any cells below) to start up **Mechanical**. Select **mm-kg-N-s** unit system. #

10.3.3 Perform Buckling Analysis

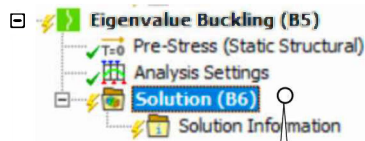


[1] Highlight **Analysis Settings of Eigenvalue Buckling**. ↓

[2] Type 2 (default) for **Max Modes to Find**. ↓



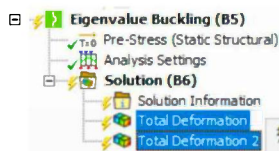
[3] Select **No** for **Include Negative Load Multiplier**. We're not interested in negative load multipliers. ↩



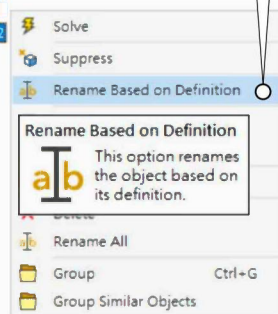
[4] Highlight **Solution of Eigenvalue Buckling.** ↓



[5] Insert **Deformation/Total.**
Do it once more. ↗

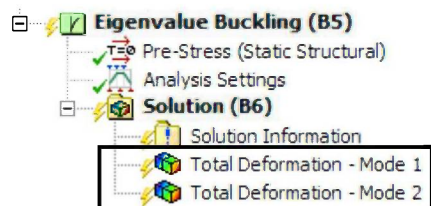


[8] Select the two result objects and right-click-select **Rename Based on Definition.** ↓

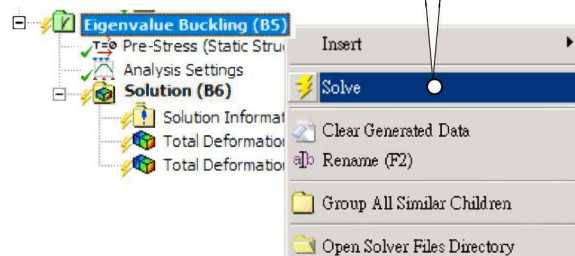


[10] Highlight **Eigenvalue Buckling** and click **Solve**. Since pre-stress is used in the buckling analysis, **Static Structural** is automatically solved before **Eigenvalue Buckling.** ↓

[11] An alternative way to solve is right-clicking **Eigenvalue Buckling** and selecting **Solve.** #



[9] The names change. ↗

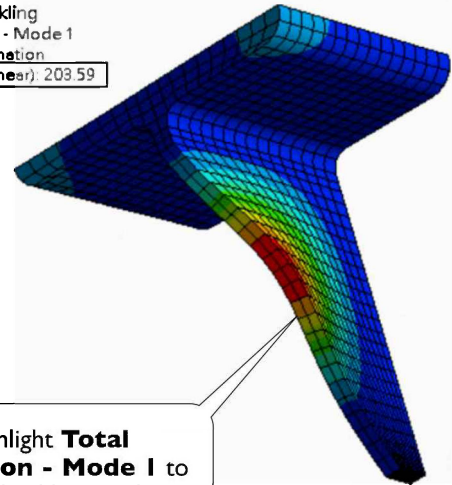


10.3.4 View the Results

[2] The **Load Multiplier** can be viewed as a safety factor as far as buckling is concerned. It predicts that 204 times of design load will cause a buckling. We conclude that the structure won't buckle under the design loads. ↓

B: Eigenvalue Buckling
Total Deformation - Mode 1
Type: Total Deformation
Load Multiplier (Linear): 203.59
Unit: mm

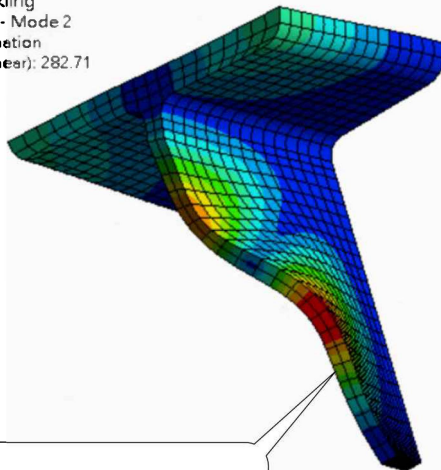
1.0009 Max
0.88973
0.77851
0.6673
0.55608
0.44486
0.33365
0.22243
0.11122
0 Min



[1] Highlight **Total Deformation - Mode 1** to view the first buckling mode. ↗

B: Eigenvalue Buckling
Total Deformation - Mode 2
Type: Total Deformation
Load Multiplier (Linear): 282.71
Unit: mm

1.0006 Max
0.88941
0.77823
0.66706
0.55588
0.4447
0.33353
0.22235
0.11118
0 Min



[3] Highlight **Total Deformation - Mode 2** to view the second buckling mode. ↓

Wrap Up

[4] Save the project and exit Workbench. #

Section 10.4

Review

10.4.1 Keywords

Choose a letter for each keyword, from the list of descriptions

1. () Eigenvalue Buckling Analysis
2. () Nonlinear Buckling Analysis
3. () Stress Stiffening Effects

Answers:

1. (B) 2. (C) 3. (A)

List of Descriptions

(A) A slender or thin structural member's bending stiffness increases with increasing axial tensile stress. Similarly, the member's bending stiffness decreases with increasing compressive stress. The extra (or deficient) stiffness is called the stress stiffness.

(B) Prediction of buckling loads and buckling modes based solely on initial stress stiffening effect. It doesn't account for large deformation effect.

(C) Prediction of buckling loads using a nonlinear analysis technique. It accounts for all nonlinear effects, including large deformation effect and nonlinear material effect.

10.4.2 Additional Workbench Exercises

Linear Buckling Analysis with Constant Loads

In the buckling analysis at the end of Section 10.1, we didn't include the lateral load (0.1 N/mm). You may wonder if the existence of the lateral load would alter the linear buckling load. Our engineering intuition tells us that a little lateral load would significantly decrease the buckling load. In linear buckling analysis, the large deformation effect is not considered. As a result, the lateral load has limited influence on the buckling loads. In our case, since the lateral load is so small, the buckling load predicted by a linear buckling analysis will be essentially the same regardless of the presence of the lateral load. This exercise requires you to verify this point.

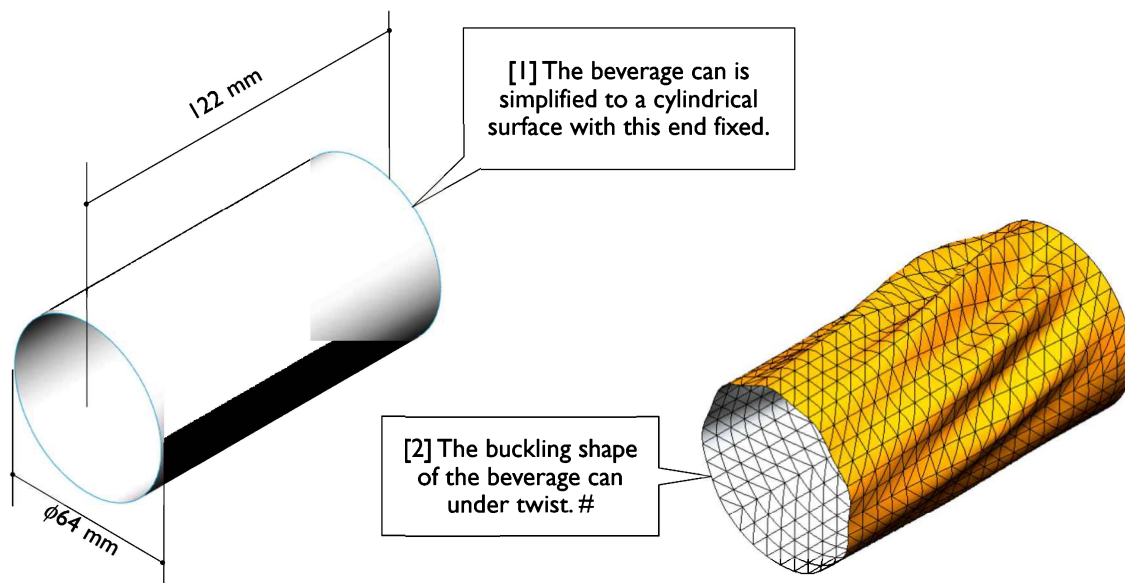
To do this, you are performing a linear buckling analysis with constant loads. Remember that a linear buckling analysis reports a multiplier. Buckling loads are ALL the applied loads multiplied by the multiplier. If the model involves a constant load, then there is no sense to multiply that constant load unless the multiplier equals a unity. Iterating on buckling analysis and trying to obtain a multiplier of unity is exactly the approach you should use when performing a linear buckling analysis with constant loads.

Buckling Pressure of Bellows Joints

The bellows joint, simulated in Section 6.1, is subject to external pressure when used in the deep ocean. If the external pressure is larger than the internal pressure, then its stability must be checked. This exercise asks you to predict the net pressure (difference between external and internal pressures) that causes the bellows joint to buckle. Also study its buckling modes.

Buckling Torque of a Beverage Can

Applying a twist on a thin beverage can introduce tensile stress on a principal direction and compressive stress on another principal direction. Excess compressive stress may cause the skin to buckle. Model a beverage can as a cylindrical surface of length 122 mm, a diameter of 64 mm, and a thickness of 0.1 mm [1]. Assume that the can is made of AA3004, which has a Young's modulus of 68.9 MPa and a Poisson's ratio of 0.35. Predict the torque that causes the skin to buckle. Study the buckling modes [2].



Chapter 11

Modal Analysis

When a structure deforms (e.g., vibrates) very fast, dynamic effects must be included in the governing equations. Modal analysis is a special type of dynamic simulation, which explores the behavior of free vibrations, vibrations without external forces. The most important characteristics of free vibrations are the natural frequencies and their corresponding vibration mode shapes.

Purpose of This Chapter

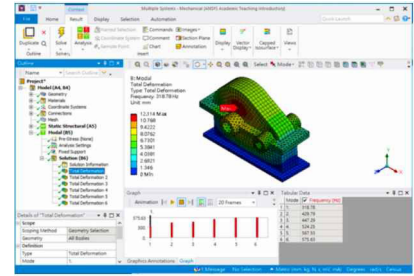
Why do we need to know the natural frequencies and the mode shapes? First, modal analysis has many usages in its own right; for example, to find the consonance frequencies of a structure in order to avoid or exploit them. This chapter provides several examples to demonstrate the usage of modal analyses. Second, these dynamic characteristics are important for further dynamic simulations; for example, transient dynamic simulations or harmonic response analyses. Chapter 12 will have several examples to demonstrate this.

About Each Section

Section 11.1 uses the gearbox, introduced in Section 6.3, as an example to illustrate a common consideration when designing a machine involving rotatory parts: avoiding resonance. Section 11.2 uses the two-story building, introduced in Section 7.3, to demonstrate the use of modal analysis to find the weakest direction of a structure and improve the stiffness of that direction. Section 11.3 discusses a case in the popular TV series *Mythbusters*, in which they succeeded in shattering CDs with a high rotational speed. Some people may believe the myth that the shattering is due to the excessive centrifugal stress. This section is designed to bust that myth, by proving that the shattering is due to resonant vibrations rather than the centrifugal stress. Section 11.4 discusses the physics of music, which is closely related to modal analysis, using a guitar string as an example.

Section 11.1

Gearbox



11.1.1 About the Gearbox

[1] In Section 6.3, we performed a static structural simulation for a gearbox. Deformation and stresses seem within safety margin under the static design loads. Dynamic behavior, however, should also be investigated.

The gearbox is designed for a speed reducer. The maximum speed at input is 630 rpm (10.5 Hz). The gearbox must be stiff enough such that, during the operation, resonance does not occur. Resonance, in this case, is harmful because it enlarges the deformation and the stresses. It also causes noises.

In this section, we want to perform a modal analysis to investigate the natural frequencies of the gearbox, to make sure these natural frequencies are much higher than the operational frequency (630 rpm).

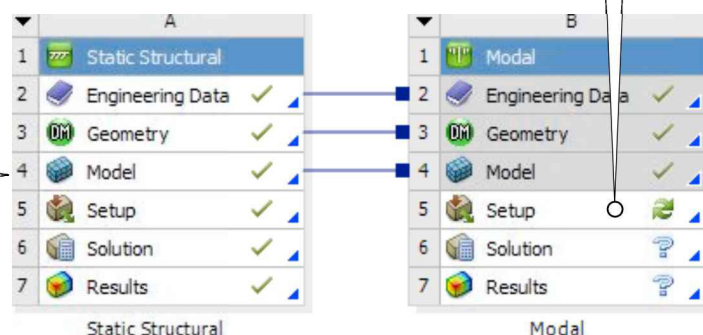
You may raise a question: when evaluating the natural frequencies, whether the design bearing loads (6.3.7[1], page 261) should be applied or not. In general, the answer is yes, since the prestress will modify the stiffness (called the stress stiffening effect, see Section 10.1). In the old days, since a "prestressed modal analysis" was expensive to perform, engineers tended to neglect the prestress effect when, according to their experiences, they knew that the prestress effect was negligible.

In this section, we will perform modal analysis twice, one with prestress, another without, to show that the effect of the prestress in this case is negligible. Note that, in other cases, the effect of prestress may be significant. #

11.1.2 Resume the Project **GearBox**

[1] Launch Workbench. Open the project **Gearbox**, which was saved in Section 6.3. ↓

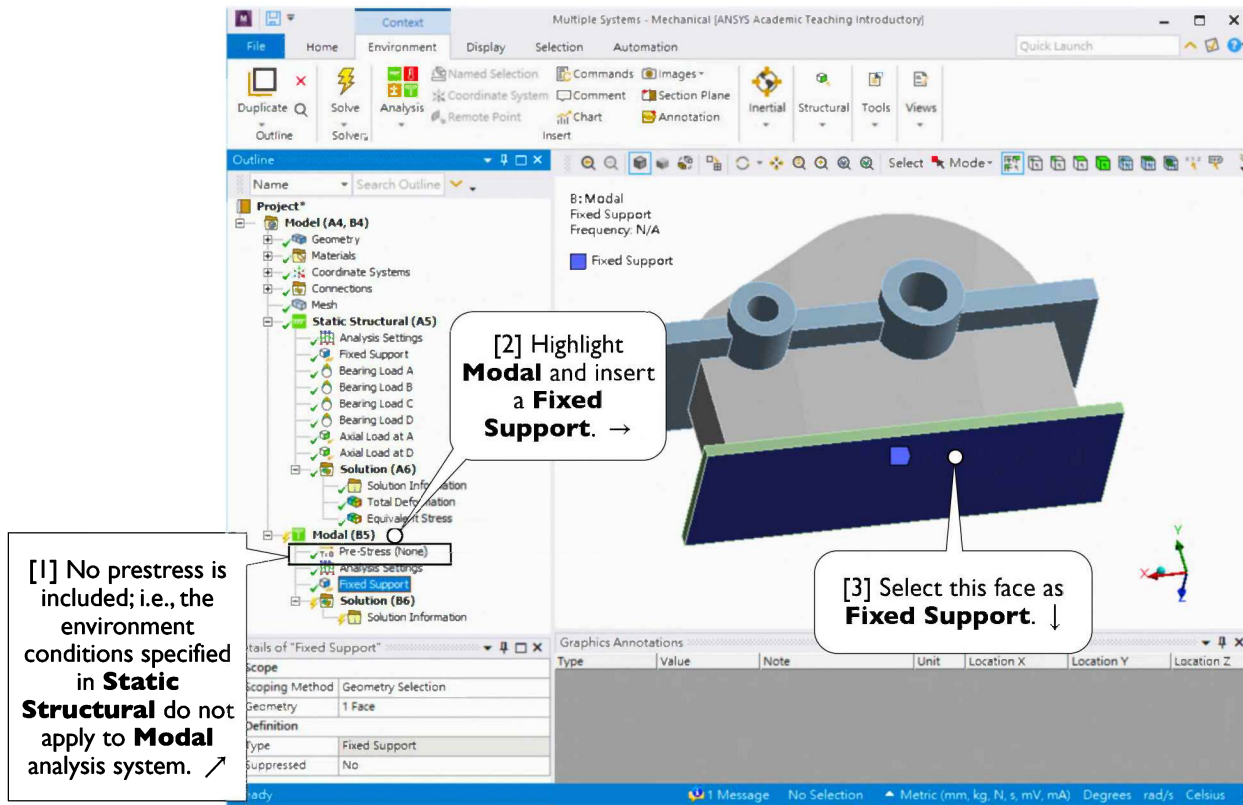
[2] From the **Toolbox**, drag the **Modal** analysis system and drop here to create an unprestressed modal analysis system. →



Static Structural analysis is not always needed for a modal analysis

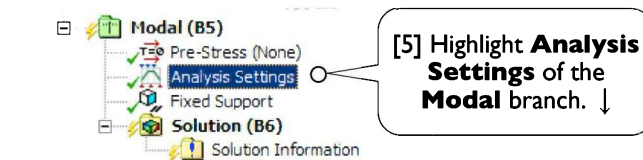
[4] We are now performing an "unprestressed modal analysis." When performing an unprestressed modal analysis, a **Static Structural** analysis is not needed (i.e., you might delete it). We keep **Static Structural** system because we want to perform a prestressed modal analysis later (11.1.5, pages 393-394). #

11.1.3 Perform Modal Analysis



Drag-and-Drop a Tree Object

[4] Steps [2-3] can be replaced by dragging the **Fixed Support** of **Static Structural** and dropping to the **Modal** branch. ←



Details of "Analysis Settings"

Options	
Max Modes to Find	6
Limit Search to Range	No
Solver Controls	
Damped	No
Solver Type	Program Controlled
Rotordynamics Controls	
Output Controls	
Analysis Data Management	

[6] The default **Max Modes to Find** is 6. Leave it unchanged. →



[7] Make sure **Modal** or any of its subordinate objects is highlighted and click **Solve**. #

11.1.4 View the Results

[4] Click **Solve**.

[1] Highlight **Solution** of the **Modal** environment. →

[2] Click **Frequency** to select all the frequencies. ↓

[3] Right-click-select **Create Mode Shape Results**.

[5] Click each result object to view a mode shape. ↓

[6] And animate the mode shape. ←

Details of "Solution (B6)"

Mode	Frequency [Hz]
1	318.76
2	429.11
3	446.47
4	523.77
5	566.61
6	571.01

Details of "Total Deformation"

Mode	Frequency [Hz]
1	318.76
2	429.11
3	446.47
4	523.77
5	566.61
6	571.01

Free Vibration Mode Shapes

[7] The vibrations you observed in [5-6] (last page) are called *free vibrations* since no external forces are applied on the structure. Like the buckling mode shapes discussed in the last chapter, values of deformation (and the corresponding stress, strain) have no physical significance; it is the shape of a vibration mode that is meaningful.

Natural Frequencies

The frequencies corresponding to the free vibrations are called *natural frequencies*. The lowest natural frequency is called the *fundamental natural frequency*, or simply *fundamental frequency*. In this case, the fundamental frequency is 318.89 Hz (19,133 rpm), far beyond the operational frequency (630 rpm). Before we jump to conclude that the resonance is not an issue, let's make sure that the prestress is negligible.

Rigid Body Modes

When performing a modal analysis, you should provide enough supports to avoid any rigid body motions. If you didn't, then rigid body modes would be included in the results. Rigid body modes have infinite period, or, equivalently, zero frequencies. Rigid body modes are superfluous, and should not be present (see 10.1.4[5], page 370). #

11.1.5 Perform Prestressed Modal Analysis

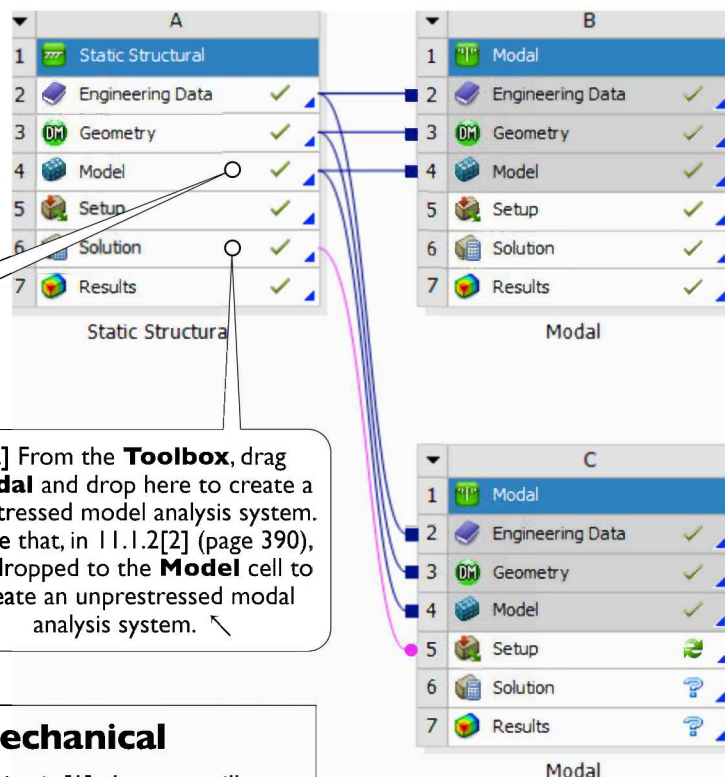
[1] Close **Mechanical**. ↘

[3] Double-click **Model** (or any cell below **Model**) to start up **Mechanical**. ↓

[2] From the **Toolbox**, drag **Modal** and drop here to create a prestressed modal analysis system. Note that, in 11.1.2[2] (page 390), we dropped to the **Model** cell to create an unprestressed modal analysis system. ↘

Actually, you don't have to close **Mechanical**

[4] If you leave **Mechanical** open without closing it [1], then you will see the project tree updated right after step [2]. ↵



[6] Turn on **Large Deflection** (see 10.1.7[3], page 372). ↓

[7] Highlight **Modal 2**. →

[8] Click **Solve**. ✓

[5] Now, **Static Structural** is included as the prestress; i.e., all the boundary conditions specified in **Static Structural** are automatically applied to **Modal 2**. More precisely, the stiffness matrix calculated in the **Static Structural**, which includes a stress stiffening effect, will be used in **Modal 2**. Stiffness and mass matrices are the only information needed for a modal analysis. ↑

[9] Highlight **Solution** of **Modal 2**. ↘



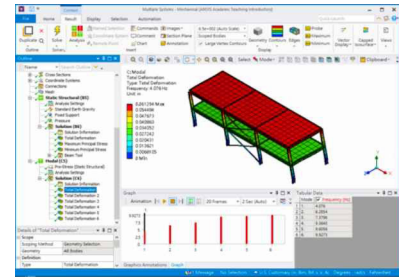
[10] The results show that the frequencies have limited differences from those in 11.1.4[2] (page 392), implying that the prestress effect may be negligible. ✓

Wrap Up

[11] Save the project and exit Workbench. #

Section 11.2

Two-Story Building



11.2.1 About the Two-Story Building

[1] In Section 7.3, we performed a static structural simulation for a two-story building. Under the static loads, the deformation and stresses are within the safety margin. Dynamic behavior, however, should be investigated for a case like this.

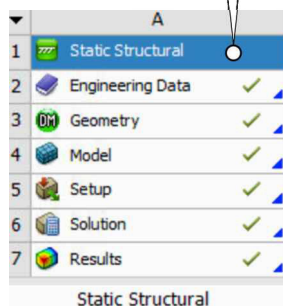
A structure's fundamental natural frequency is proportional to the structure's stiffness: the higher the frequency, the stiffer the structure. The stiffnesses of two structures can be compared using their fundamental frequencies. For a structure, stiffnesses in different directions can be compared using the lowest frequencies in the respective directions. When we want to reinforce a structure, we should first reinforce the direction which has the lowest frequency.

Local building codes usually require a minimum frequency so that a building would not be too soft (i.e., not stiff enough), both for comfort and safety concerns. In this section, we will perform modal analyses using the two-story building model. Prestress is included in the simulation. We will find that the fundamental frequency of the building is too low. We then propose a simple solution to improve the stiffness of the building. #

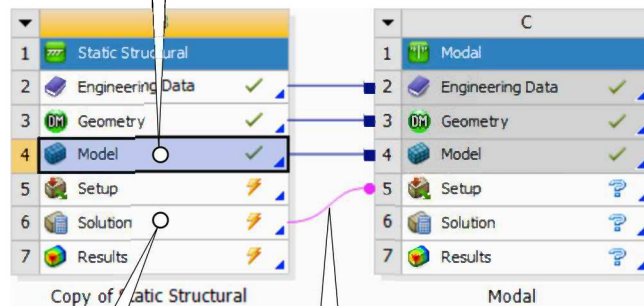
11.2.2 Resume the Project **Building**

[1] Launch Workbench. Open the project **Building**, which was saved in Section 7.3. ✓

[2] Right-click here and select **Duplicate**. We will use the duplicated system in this section and return to this system in Section 12.3. ↓



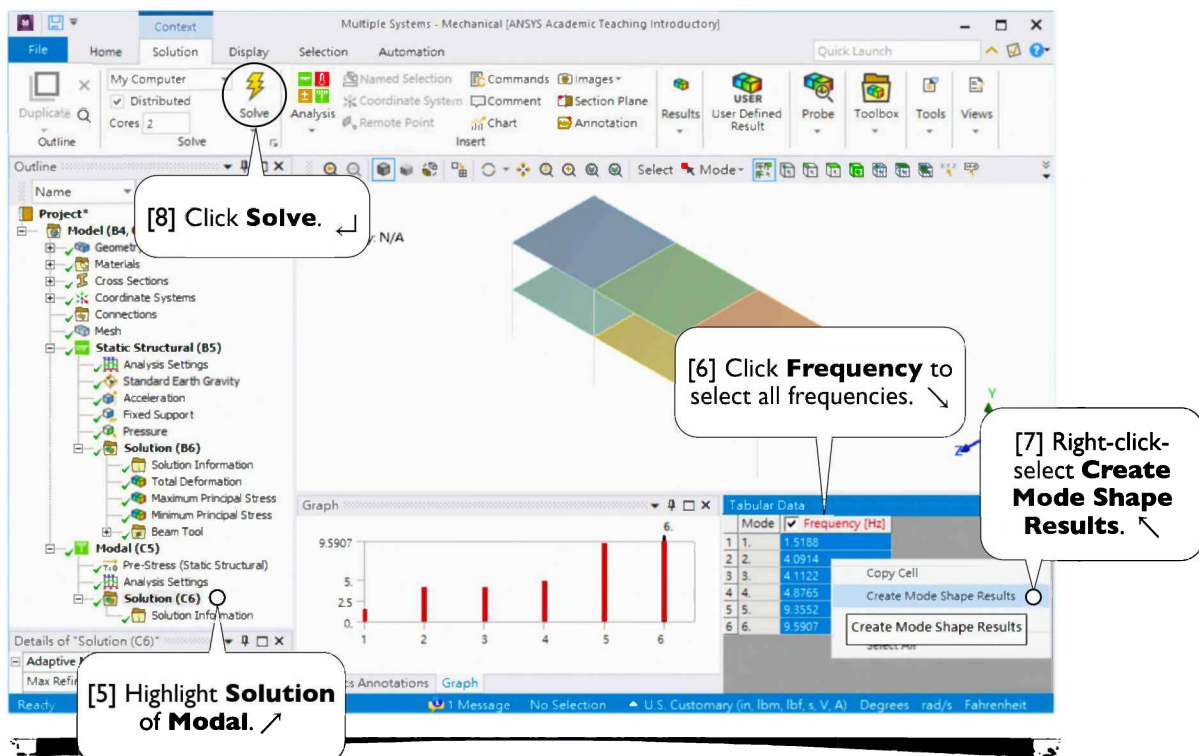
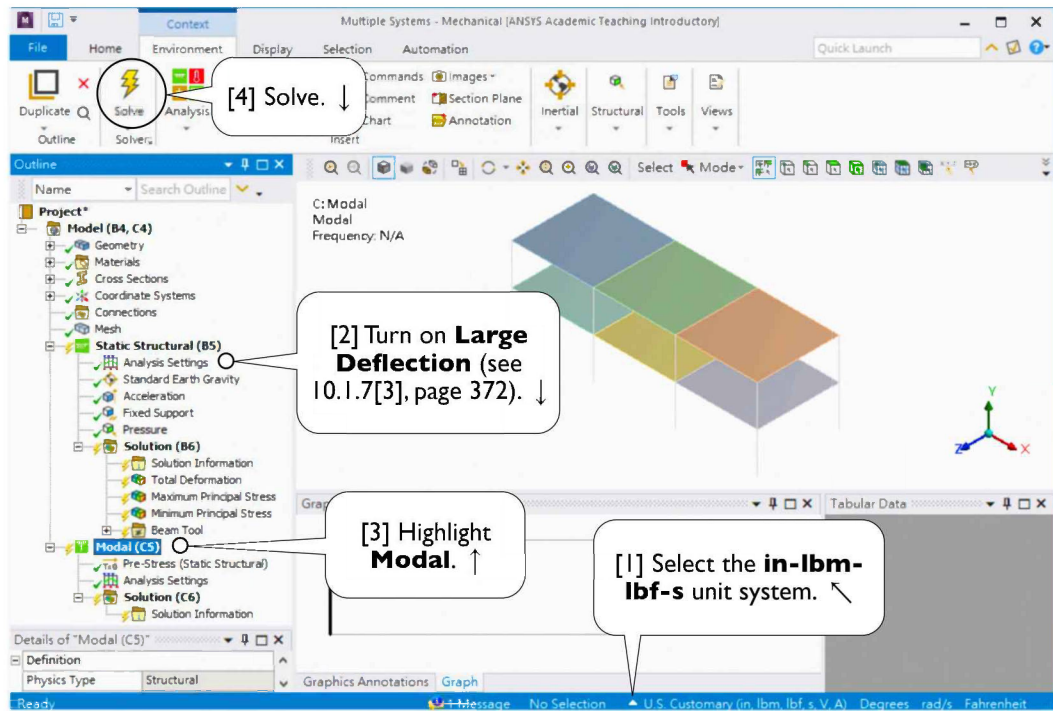
[5] Double-click **Model** (or any cell below) to start up **Mechanical**. #

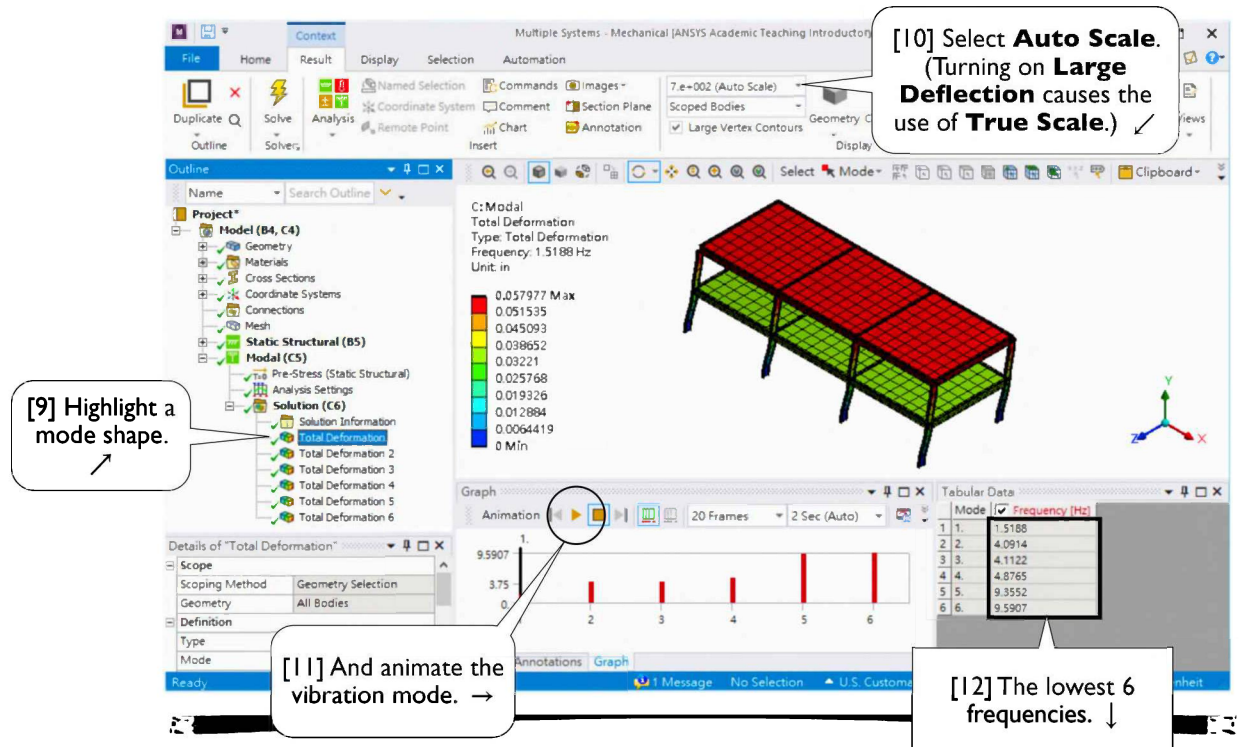


[3] From **Toolbox**, drag the **Modal** analysis system and drop to **Solution** of the new system. →

[4] A link with a circular connector means that the data is transferred to the new cell; i.e., the results of static analysis are used as a prestress for the modal analysis. ↑

11.2.3 Perform Modal Analysis





Discussion of the Vibration Modes

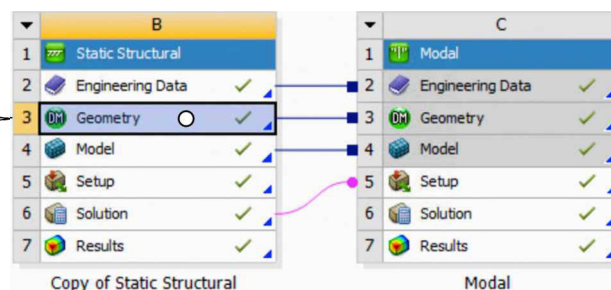
[13] The fundamental frequency is 1.52 Hz, a vibration in X-direction (this can be observed from the animation). The second lowest frequency is 4.09 Hz, a vibration in Z-direction. The third lowest frequency mode is also in X-direction. The fourth lowest frequency mode is a torsional vibration in XZ-plane. The fifth and sixth modes are vertical (Y-direction) vibrations of the floors.

A frequency of 1.52 Hz is not only uncomfortable but also unsafe. Imagine a group of young people dancing on the building's floor. The rhythmic loading of the floor may cause a safety issue, since the tempo of the music is possibly close to the building's fundamental frequency. A harmonic response analysis will be conducted to clear up this safety issue in Section 12.3. Some local building codes require a structure's natural frequency be larger than 5 Hz if the structure is to be used as a dance floor in a venue.

A simple remedy for the building is to add diagonal members to the weakest direction. Location of the diagonal members should be carefully chosen to avoid conflicting with the building's architectural functionalities. #

11.2.4 Modify the Geometry

[1] Close **Mechanical** and double-click **Geometry** to modify the geometry. ↩



[2] Pull-down-select **Concept/Lines From Points**.

[3] Create 8 line segments. This wireframe display is made by turning off **View/Cross Section Solids**.

[4] Click **Apply**.

[5] Select **Add Frozen**. The lines are created as separate parts because we want to specify a different cross section.

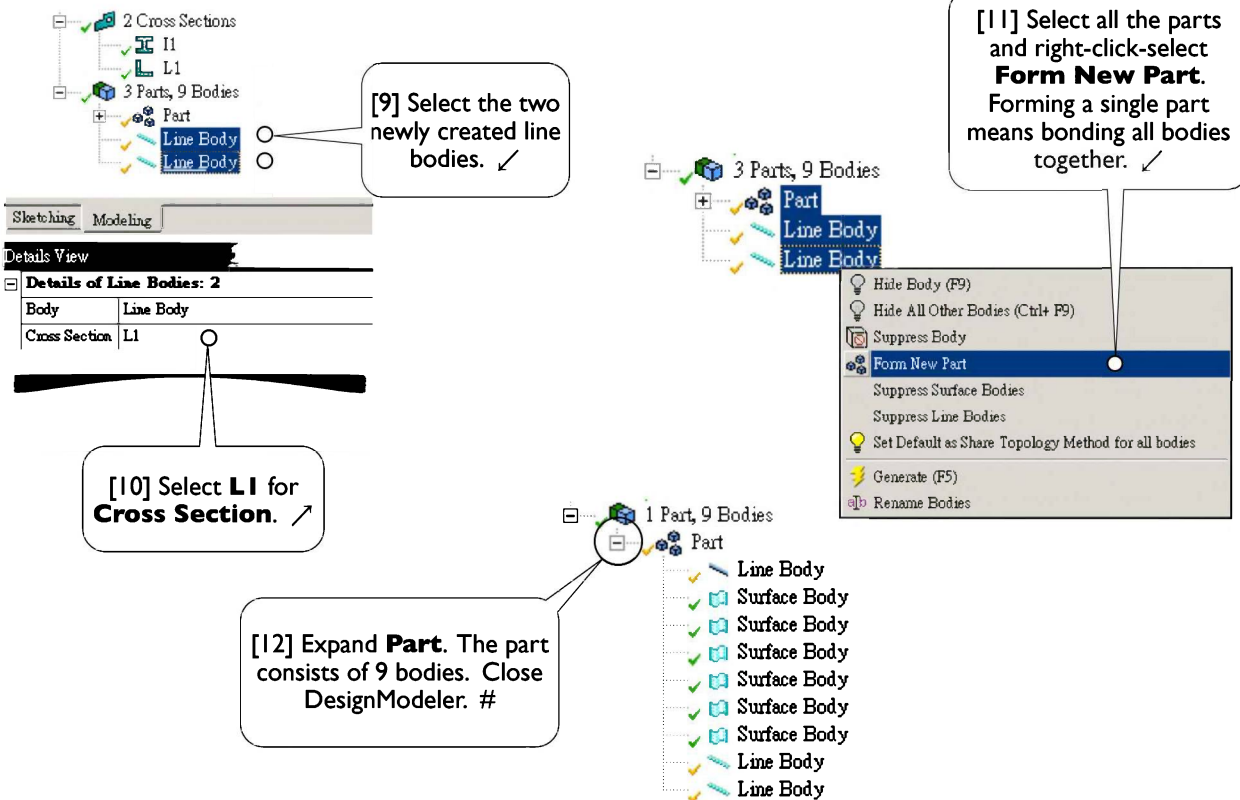
[6] Click **Generate**.

[7] Pull-down-select **Concept/Cross Section/L Section**.

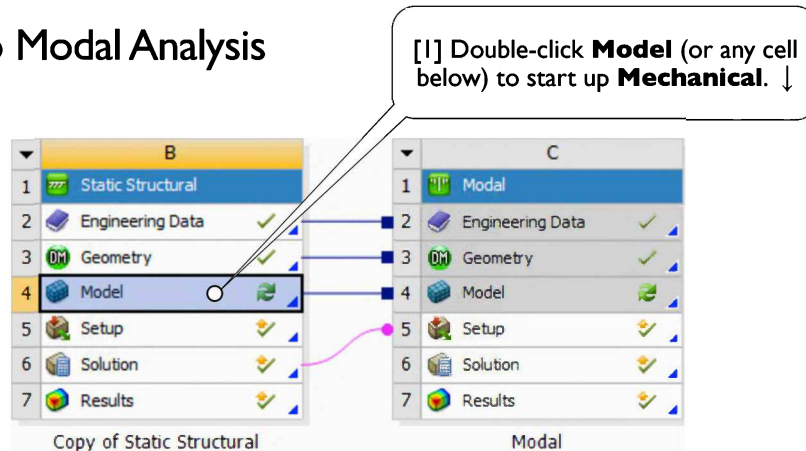
[8] Type dimensions for the L4x4x1/2 angle.

Details of Line3	
Lines From Points	Line3
Point Segments	8
Operation	Add Frozen

Details of L1	
Sketch	L1
Show Constraints?	No
Dimensions: 4	
W1	4 in
W2	4 in
t1	0.5 in
t2	0.5 in

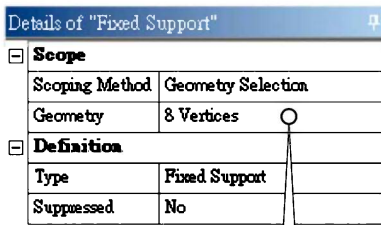


11.2.5 Return to Modal Analysis



Check the environment conditions every time you modify geometry

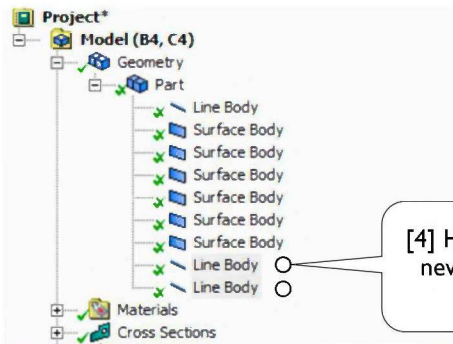
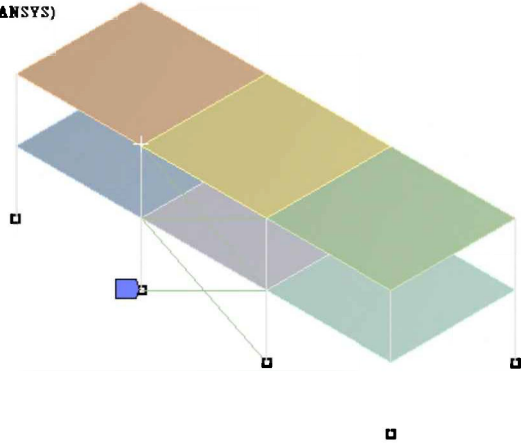
[2] After you modify the geometry, always check environment conditions such as loads and supports. In this case, some of the fixed supports are lost; we need to fix them (see [3], next page). We'll delete **Acceleration** in **Static Structural** branch [4], which represents an earthquake load. We usually don't consider earthquake load as a "prestress." We do consider earth gravity as prestress. However, the effect of the prestress on natural frequencies is very limited. This is left as an exercise (11.5.2, page 418). ↵



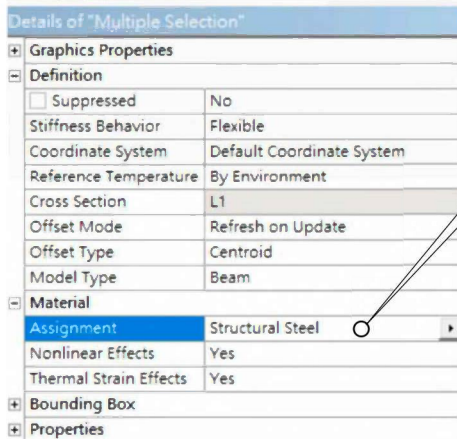
[3] Highlight **Fixed Support** in the **Static Structural** environment, reselect the 8 vertices at the column bases, and click **Apply**. Turn on vertex selection filter if needed. →

A: Static Structural (ANSYS)
Fixed Support
Time: 1. s

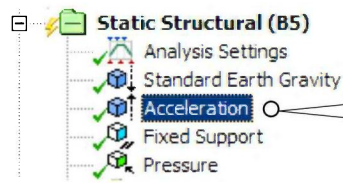
Fixed Support



[4] Highlight the two newly created line bodies. ↓



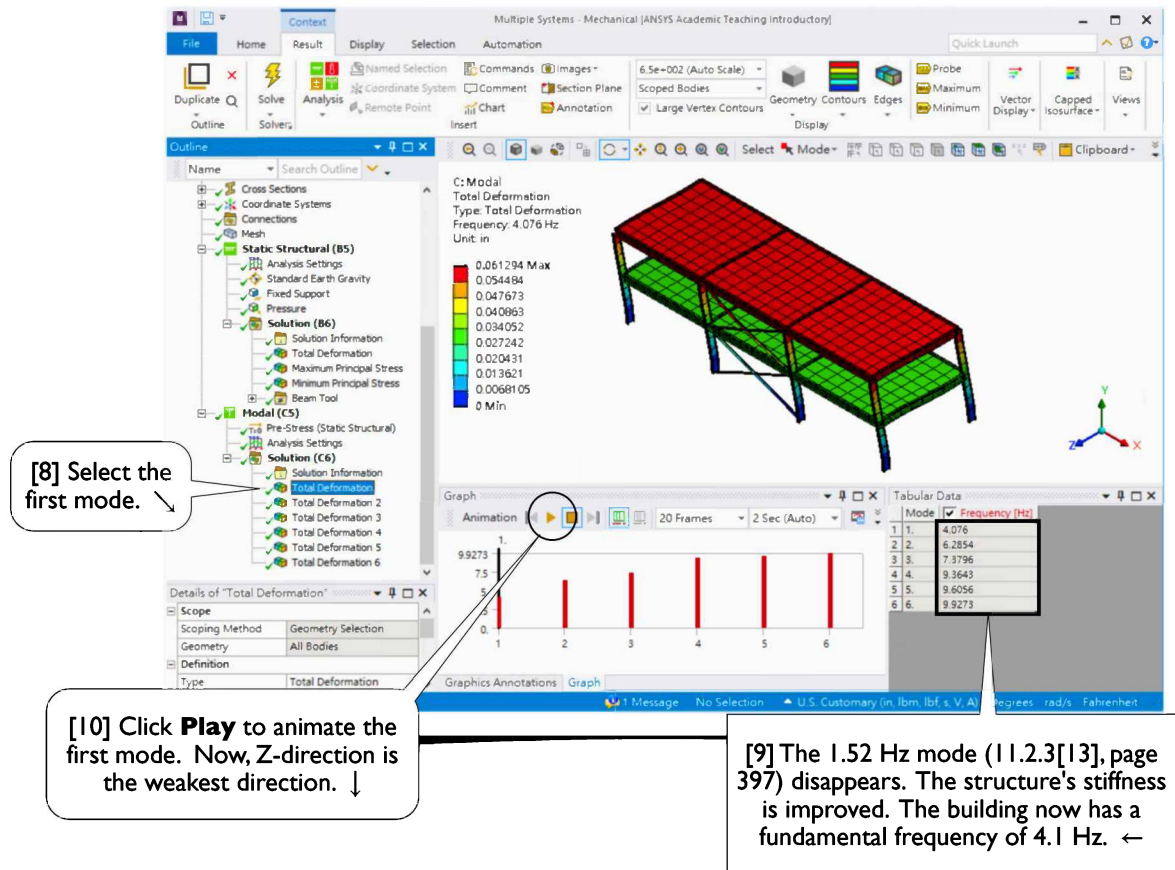
[5] And select **Structural Steel** for the material assignment. ↓



[6] Right-click **Acceleration** and select **Delete**. Note that, in 11.2.3, we could have deleted **Acceleration**; we didn't do that because the results would have almost no differences. ←



[7] Highlight **Modal** and click **Solve**. ↩



Discussion

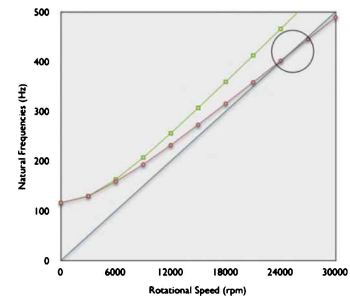
[11] After the reinforcement of X-direction, Z-direction now becomes the weakest direction [10]. If we want to further improve the structure's stiffness, we may add bracing members in the Z-direction. This is left as an exercise (11.5.2, page 418). ↓

Wrap Up

[12] Save the project and exit Workbench. #

Section 11.3

Compact Disk



11.3.1 About the Compact Disk

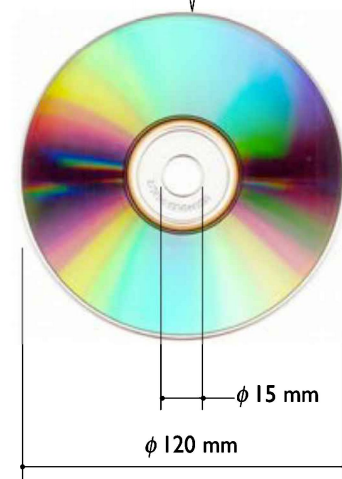
[1] A CD is made of a polycarbonate (PC) [2], with density 1200 kg/m^3 , Young's modulus 2.2 GPa , Poisson's ratio 0.37 , and tensile strength 65 MPa . The outer diameter is 120 mm , the inner (hole) diameter is 15 mm , and the thickness is 1.2 mm . For a $52\times$ CD drive, the maximum rotational speed reaches $27,500 \text{ rpm}$ (458 Hz) when reading the inner tracks^[Ref 1].

The television series *MythBusters* conducted experiments^[Refs 2-4] in which they succeeded in shattering CDs at speeds of $23,000 \text{ rpm}$. When conducting the experiments, they press the CD between two nuts, which are 27 mm in diameter^[Ref 5].

In this section, we first want to find out the maximum stress in the CD due to the centrifugal force when rotating in $27,500 \text{ rpm}$, to justify that the shattering may not be due to the centrifugal stress.

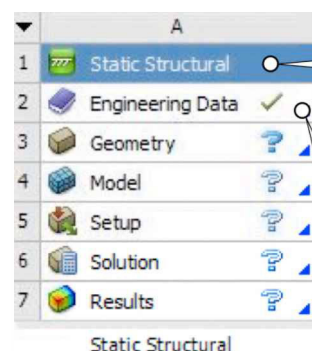
Second, we want to find the natural frequencies of the CD to investigate the possibility of resonant vibrations. We will conclude that the CD shattering may be due to sustaining vibrations rather than centrifugal stress. Also, we want to demonstrate that the natural frequencies increase with increasing rotational speed. →

[2] The CD is made of a polycarbonate (PC) plastic. #



11.3.2 Start Up a New Project

[1] Launch Workbench. Save the project as **CD**. Create a **Static Structural** analysis system [2]. Use **SI** as the project units. Double-click **Engineering Data** [3] and create a new material with the name **PC** ([4], next page). Input the material properties: a density (1200 kg/m^3), a Young's modulus (2.2 GPa), and a Poisson's ratio (0.37) [5-7]. Click **Project** to return to **Project Schematic** [8]. →



[2] Create a **Static Structural** analysis system. ↓

[3] Double-click **Engineering Data**. ↵

[8] Click **Project**. #

[5] Double-click to include **Density**. ✓

[4] Type **PC** for the new material. ←

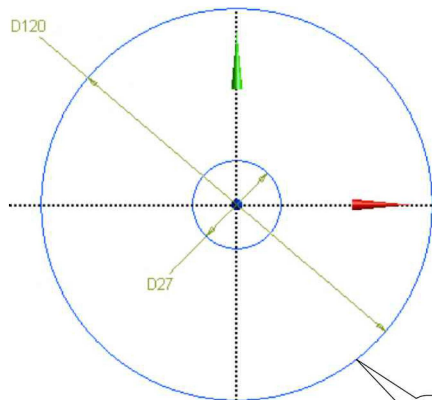
[7] Type 1200 (kg/m³) for **Density**, 2.2e9 (Pa) for **Young's Modulus**, and 0.37 for **Poisson's Ratio**. Note that the unit system is **SI**. ←

[6] Double-click to include **Isotropic Elasticity**. →

Property	Value	Unit
Density	1200	kg m ⁻³
Young's Modulus	2.2E+09	Pa
Poisson's Ratio	0.37	
Bulk Modulus	2.8205E+09	Pa
Shear Modulus	8.0292E+08	Pa

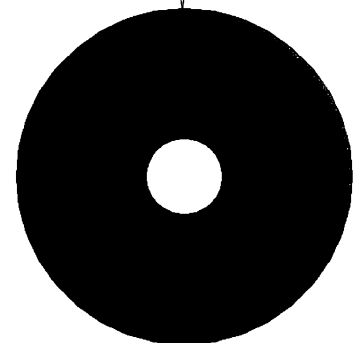
11.3.3 Create Geometry in DesignModeler

[1] Start up DesignModeler. Select **Millimeter** as the length unit. Draw a sketch on **XYPlane** as shown in [2]. Pull-down-select **Concept/Surfaces From Sketches** to create a surface body from the sketch [3]. Close DesignModeler. ↓



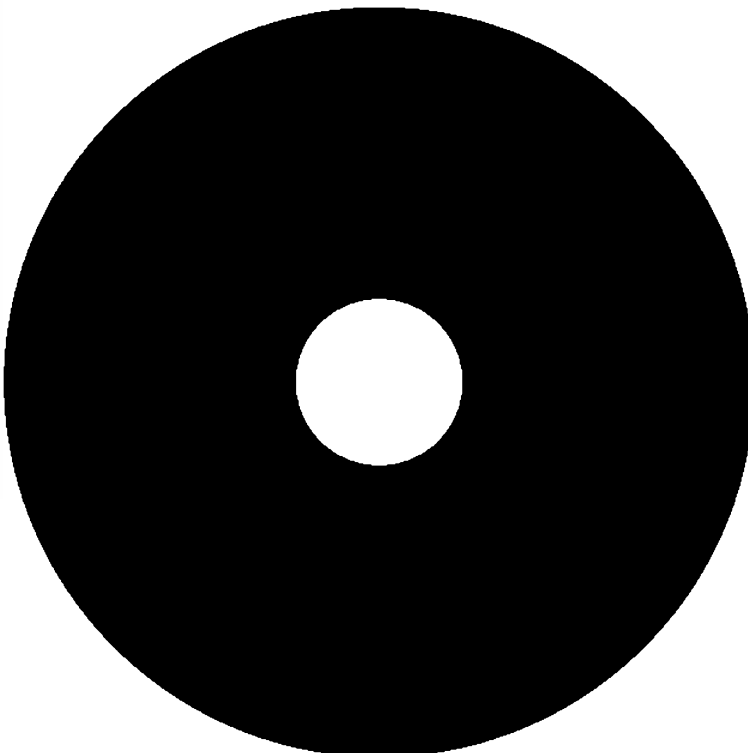
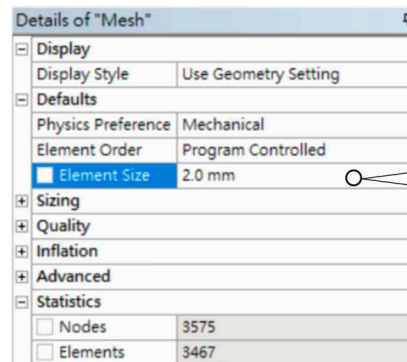
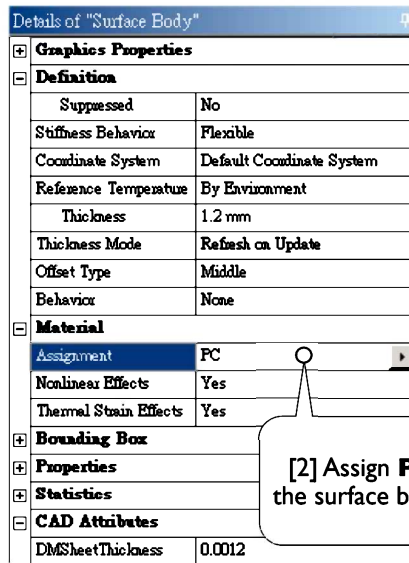
[2] Create a sketch like this. Note that the inner diameter is 27 mm, which is the diameter of the nuts pressing on the CD (see 11.3.1 [1], last page). ↗

[3] Create a surface body of thickness 1.2 mm using **Concept/Surface From Sketches**. #

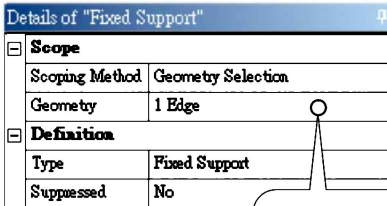


11.3.4 Assess Centrifugal Stress

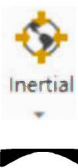
[1] Start up **Mechanical**. Select the unit system **mm-kg-N-s**. Assign the material **PC** for the surface body [2]. Specify 2.0 (mm) for the maximum element size [3] and generate mesh. ✓



[4] Select the inner rim and insert a **Fixed Support** [5]. Insert an **Inertial/Rotational Velocity** load and type 27,500 (RPM) for **Z Component** [6-7]. Insert a **Maximum Principal Stress** [8] and solve the model [9]. ✓

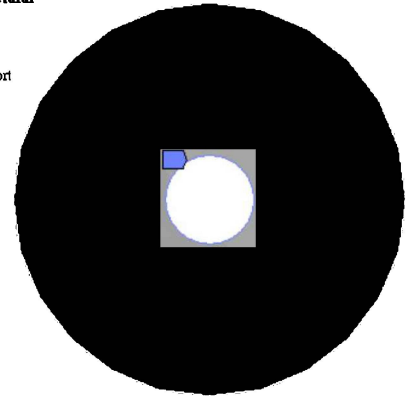


[5] Select the inner rim as **Fixed Support**. Note that the CD is pressed by two nuts which have a diameter of 27 mm (see 11.3.1[1], page 402). ↓

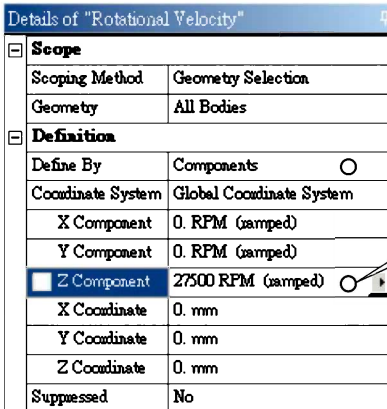
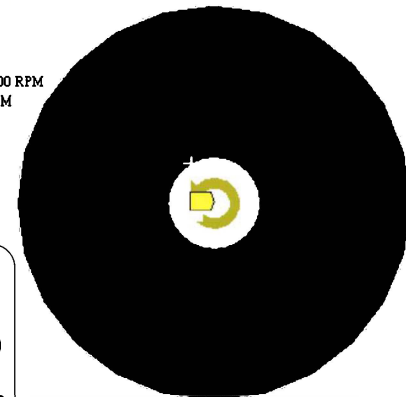


[6] Insert a **Inertia/Rotational Velocity**. ↓

A: Static Structural
Fixed Support
Time: 1. s
Fixed Support



A: Static Structural
Rotational Velocity
Time: 1. s
Rotational Velocity: 27500 RPM
Rotation: 0,0,27500 RPM
Location: 0,0,0. mm

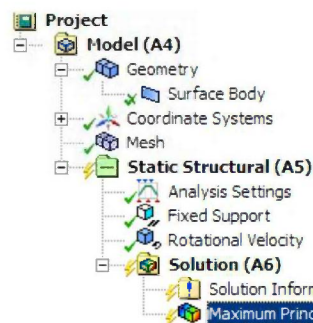


[7] Select **RPM** as the unit of angular velocity (by opening the units menu, see 3.1.5[6], page 113) and type 27500 (RPM) for **Z Component**. This applies on the entire model. ↘

✓ RPM



[9] Solve. ↵

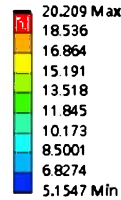


[8] Insert a **Maximum Principal Stress**. ←

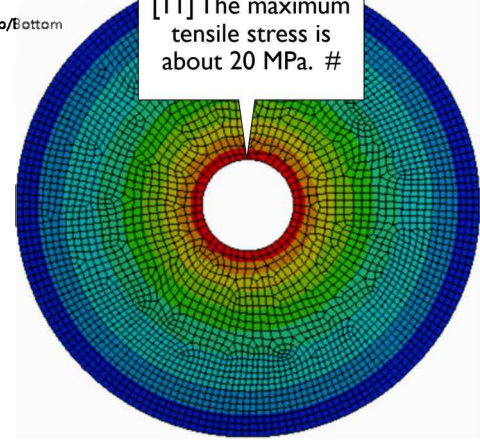
[10] The maximum tensile stress is about 20 MPa [11], which occurs at the inner rim as expected and is far less than the material's tensile strength (65 MPa). It is unlikely that the CD shatters due to the tensile stress, unless some defects already exist in the CD before the experiments.

As the next step, let's assess the CD's natural frequencies under the prestress caused by the high-speed spinning. →

A: Static Structural
Maximum Principal Stress
Type: Maximum Principal Stress - Top/Bottom
Unit: MPa
Time: 1



[11] The maximum tensile stress is about 20 MPa. #

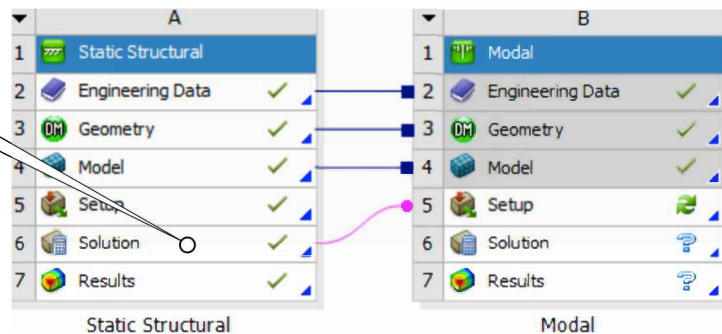


11.3.5 Assess Natural Frequencies

[1] Close **Mechanical**. From **Toolbox**, drag-and-drop the **Modal** analysis system to the **Solution** cell of **Static Structural** [2]. Start up **Mechanical** by double-clicking any cells other than **Engineering Data** or **Geometry**. Turn on **Large Deflection** [3]. Highlight **Modal** and solve the model [4]. The lowest frequency [5] is close to the maximum of the operational frequencies (2,7500 rpm, or 458 Hz), which may resonate the CD and may cause damage.

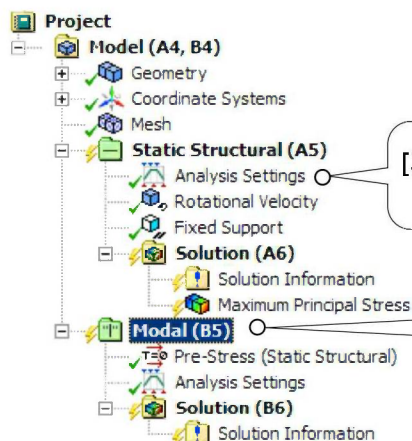
Let's look into this issue of resonant vibrations more thoroughly. We'll assess the prestressed natural frequencies for a range of possible operational speeds, from 0 to 30,000 rpm. ✓

[2] From **Toolbox**, drag-and-drop the **Modal** analysis system here. ↓



[3] Turn on **Large Deflection**. ↓

[4] Highlight **Modal** and solve the model. ↓



Tabular Data		
	Mode	Frequency [Hz]
1	1.	452.65
2	2.	528.25
3	3.	528.28
4	4.	705.24
5	5.	705.25
6	6.	927.13

[5] The results show that resonant vibrations are possible. Create mode shape results for these frequencies and animate each mode shape (remember to set to **Auto Scale**). #

11.3.6 Assess Natural Frequencies over the Range of Rotational Speeds

[1] Highlight Rotational Velocity and select Z Component as an input parameter. →

Details of "Rotational Velocity"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Define By	Components
Coordinate System	Global Coordinate System
X Component	0 RPM (gamped)
Y Component	0 RPM (gamped)
P Z Component	27500 RPM (gamped)
X Coordinate	0 mm
Y Coordinate	0 mm
Z Coordinate	0 mm
Suppressed	No

[2] Highlight Total Deformation (the first mode) and select Frequency as an output parameter. →

Details of "Total Deformation"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Total Deformation
Mode	1.
Identifier	
Suppressed	No
Results	
Minimum	0. mm
Maximum	375.58 mm
Maximum Occurs On	Surface Body
Maximum Occurs On	Surface Body
Information	
P Frequency	452.65 Hz

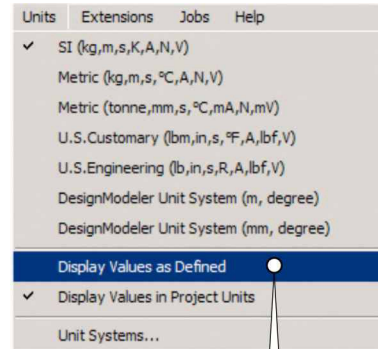
[3] Highlight Total Deformation 2 (the second mode) and select Frequency as an output parameter. ↓

Details of "Total Deformation 2"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Total Deformation
Mode	2.
Identifier	
Suppressed	No
Results	
Minimum	0. mm
Maximum	551.29 mm
Maximum Occurs On	Surface Body
Maximum Occurs On	Surface Body
Information	
P Frequency	528.25 Hz

[4] Close Mechanical and double-click Parameter Set. ↵

Table of Design Points				
	A	B	C	D
1	Name	P1 - Rotational Velocity Z Component	P2 - Total Deformation Reported Frequency	P3 - Total Deformation 2 Reported Frequency
2	Units	rev min ⁻¹	Hz	Hz
3	DP 0 (Current)	27500	452.65	528.25
4	DP 1	0.01		
5	DP 2	3000		
6	DP 3	6000		
7	DP 4	9000		
8	DP 5	12000		
9	DP 6	15000		
10	DP 7	18000		
11	DP 8	21000		
12	DP 9	24000		
13	DP 10	27000		
14	DP 11	30000		
*				

[5] Make sure the unit is **rev min⁻¹** (i.e., **RPM**; if it is not, see [6]). Type eleven additional **Design Points**. Note that we type a small value for the first design point, since zero value is not allowed. →



[6] If the unit of rotational velocity is not **rev min⁻¹** (i.e., **RPM**), turn on **Units/Display Values as Defined**. ↓

⚡ Update All Design Points

[7] Click **Update All Design Points**. ↓

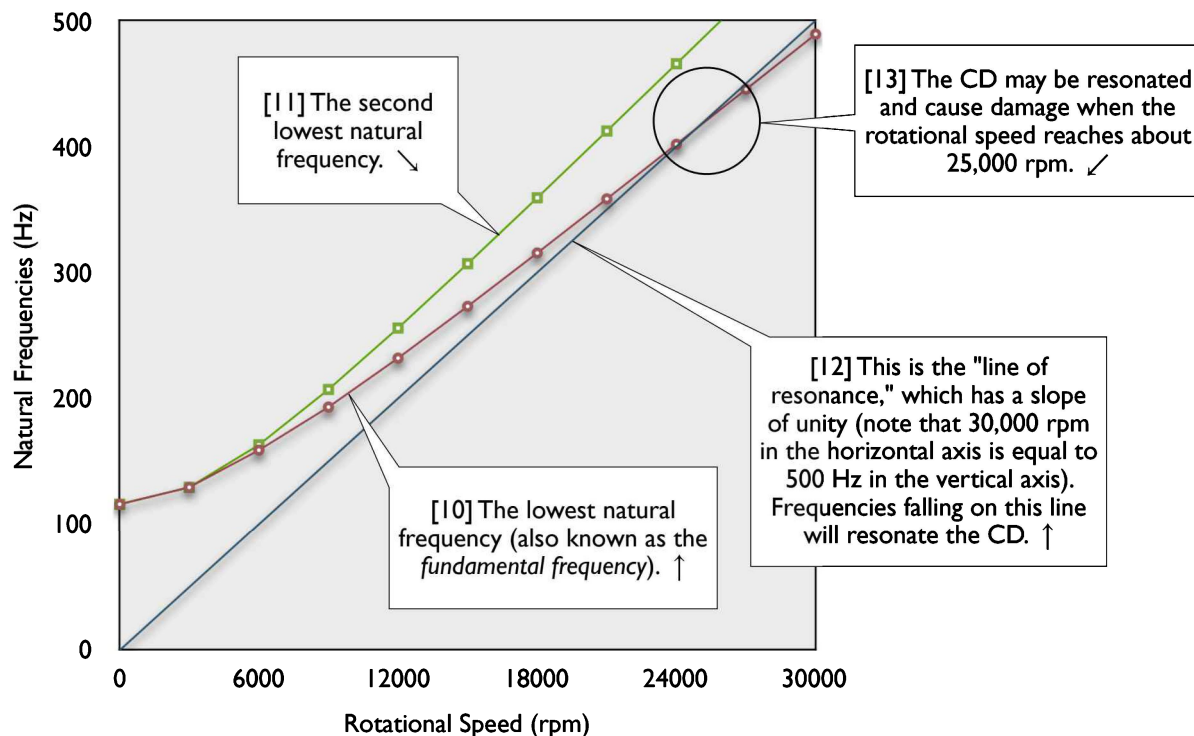
Table of Design Points				
	A	B	C	D
1	Name	P1 - Rotational Velocity Z Component	P2 - Total Deformation Reported Frequency	P3 - Total Deformation 2 Reported Frequency
2	Units	rev min ⁻¹	Hz	Hz
3	DP 0 (Current)	27500	452.65	528.25
4	DP 1	0.01	116.01	116.04
5	DP 2	3000	129.46	129.48
6	DP 3	6000	158.92	163.13
7	DP 4	9000	193.16	207.11
8	DP 5	12000	231.99	255.88
9	DP 6	15000	273.11	306.97
10	DP 7	18000	315.44	359.27
11	DP 8	21000	358.42	412.26
12	DP 9	24000	401.78	465.65
13	DP 10	27000	445.37	519.29
14	DP 11	30000	489.11	573.1
*				

[8] Now, we use these results to plot a chart on the next page. ↩

[9] The data [8] (last page) are plotted in a chart as shown [10-11], which shows that the CD may be resonated when the rotational speed reaches about 25000 rpm [12-13]. The excitation may in turn cause damage.

According to these studies, we may conclude that the CD shattering in the experiments conducted by the TV series *Mythbusters* is probably due to resonant vibration effects, rather than the centrifugal stress.

A real CD drive usually provides supports for the CD, to reduce the vibrations as well as the radial deformations; therefore, the vibrations may not occur in a real CD drive. ↓



Wrap Up

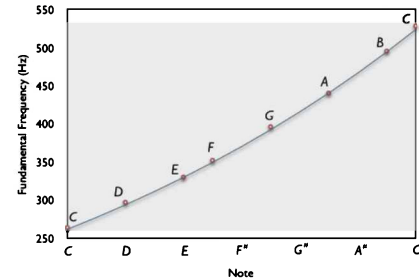
[14] Save the project and exit Workbench. #

References

1. Wikipedia>Optical disc drive
2. <https://www.youtube.com/watch?v=g7puuZlh-oM>
3. Wikipedia>MythBusters (2003 season)>Episode 2.3 Exploding CDs.
4. <https://www.youtube.com/watch?v=zs7x1Hu29Wc>
5. Acknowledgement: Thanks to Professor Per Blomqvist of the University of Gavle in Sweden, who pointed out this fact and suggested a more realistic simulation of this exercise. In the earlier versions of the book, I assumed that the CD was fixed in the inner rim, which has a diameter of 15 mm.

Section 11.4

Guitar String



This section introduces the physics of music. When designing or improving a musical instrument, an engineer must know the physics of music. Likewise, to fully appreciate the theory of music, a musician needs to know the physics behind the music.

We will use a guitar string to demonstrate some of the physics of music in this section and Section 12.5. For those students who are not interested in music theory, you may read only 11.4.1 to 11.4.3 (pages 410-413) and skip the rest of the material in this section. On the other hand, if you want to introduce this article to a friend who does not have enough background in modal analyses, he may skip 11.4.1 and 11.4.2 and jump to 11.4.3 (page 413) directly.

11.4.1 About the Guitar String

[1] The guitar string in our case is made of steel, which has a mass density of 7850 kg/m^3 , a Young's modulus of 200 GPa , and a Poisson's ratio of 0.3 . It has a circular cross section of diameter 0.28 mm and a length of 1.0 m . The string is stretched with a tension T , and is in tune with a standard A note (a), which is defined as 440 Hz in modern music. In 11.4.2, we will perform a modal analysis to find the required tension T .

Before performing the simulation, let's make some simple calculations. According to physics, the wave traveling on a string has a speed of

$$v = \sqrt{\frac{T}{\mu}} \quad (1)$$

Where μ is the linear density (kg/m) of the string. The standing wave corresponding to the lowest frequency is called the *first harmonic mode*, which has a wavelength of twice the string length ($2L$). According to the relation between the velocity, the frequency, and the wavelength,

$$f = \frac{v}{\lambda} = \frac{v}{2L} \quad (2)$$

According to (1) and (2), we can estimate the required tension,

$$T = \mu(2fL)^2 = 7850 \times \frac{\pi(0.00028)^2}{4} (2 \times 440 \times 1.0)^2 = 374.32 \text{ N} \quad (3) \quad \#$$

11.4.2 Perform Modal Analysis

[1] Launch Workbench. Save the project as **String**. Create a **Static Structural** system. Drag-and-drop the **Modal** analysis system to the **Solution** cell of the **Static Structural** system. In **Engineering Data**, make sure the material properties for **Structural Steel** are consistent with those of the guitar string given in 11.4.1 (a mass density of 7850 kg/m^3 , a Young's modulus of 200 GPa , and a Poisson's ratio of 0.3).

Start up DesignModeler. Select **Millimeter** as the length unit. On **XYPlane**, draw a line of 1000 mm on X-axis. Create a line body from the sketch. Create a circular cross section of radius 0.14 mm , and assign the cross section to the line body. Close DesignModeler. ↵

[2] Start up **Mechanical**. Make sure the unit system is **mm-kg-N-s**. Under **Static Structural**, specify environment conditions [3]: a **Fixed Support** [4], a **Displacement** [5], and a **Force** [6]. Now, there should be no rigid body modes. ✓

A: Static Structural
Static Structural
Time: 1. s

Force: 374.32 N
Displacement
Fixed Support

[3] The environment conditions (see [4-6]). ↓



Details of "Fixed Support"	
Scope	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Fixed Support
Suppressed	No

[4] At the left end, all 6 degrees of freedom are fixed. ↓

[7] In **Analysis Settings of Modal**, type 12 for **Max Modes to Find**. ↓

Details of "Displacement"	
Scope	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
Y Component	0. mm (amped)
Z Component	0. mm (amped)
Suppressed	No

[5] At the right end, the Y- and Z-translational degrees of freedom are fixed. ↓

Details of "Analysis Settings"	
Options	
Max Modes to Find	12
Limit Search to Range	No
Spin Softening	Program Controlled
Solver Controls	
Damped	No
Solver Type	Program Controlled
Rotodynamic Controls	
Output Controls	
Analysis Data Management	

Details of "Force"	
Scope	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	374.32 N (amped)
Y Component	0. N (amped)
Z Component	0. N (amped)
Suppressed	No

[6] At the right end, apply a force of 374.32 N (see Eq. 11.4.1(3), last page). ✓

Tabular Data		
	Mode	Frequency [Hz]
1	1.	432.92
2	2.	432.92
3	3.	800.24
4	4.	865.85
5	5.	865.85
6	6.	1242.
7	7.	1298.8
8	8.	1298.8
9	9.	1731.9
10	10.	1731.9
11	11.	2165.1
12	12.	2165.1

[8] Turn on **Large Deflection** (11.3.5[3], page 406). Perform a modal analysis. (Remember to highlight **Model**.) ↓

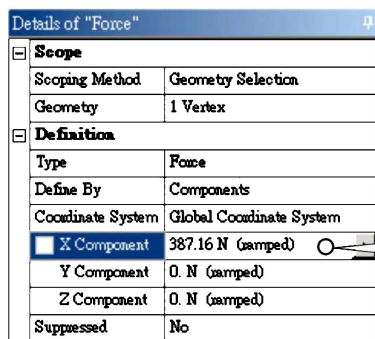
[9] The results of the modal analysis [8] show that the lowest frequency is not exactly 440 Hz. It slightly deviates from what we've predicted using Eq. 11.4.1(3), last page. The main reason is that when applying the tension, the length of the string increases (about 30 mm). Another reason is due to the use of beam model, which is slightly different from the pure tension-only model used in our hand-calculation. ✓

[10] After several trial-and-errors (try this on your own), we come up with a tension of 387.16 N [11] that exactly produces a fundamental frequency of 440 Hz [12].

As expected, the frequencies [12] include 440 Hz, 880 Hz, 1320 Hz, 1760 Hz, 2200 Hz (which are all integral multiplications of the fundamental frequency, 440 Hz), with negligible numerical errors. Before we discuss these "harmonic modes" in 11.4.3 (next page), let's take a look at those non-harmonic modes first.

The third mode (800.8 Hz) is a rotation mode in X-direction [13-14]. In the real-world, it exists, but its magnitude is usually very small, and our ears hardly sense it since the air pressure is hard to be excited in this way.

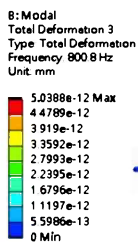
The sixth mode (1241.3 Hz) is a stretching mode [15], visible in an animation. In the real-world, it exists too, but again, we hardly sense it since the air pressure is hard to be excited in this way. ✓



[11] Change the force to 387.16 N. Solve again. →

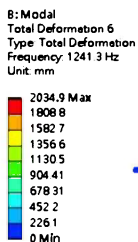
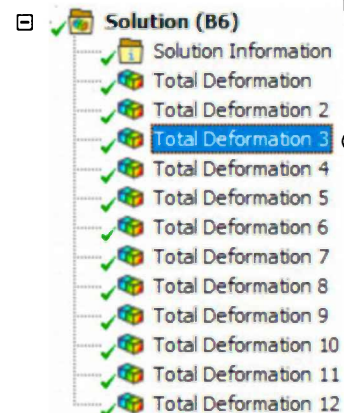
Tabular Data		
	Mode	Frequency [Hz]
1	1.	440.
2	2.	440.
3	3.	800.8
4	4.	880.
5	5.	880.
6	6.	1241.3
7	7.	1320.
8	8.	1320.
9	9.	1760.2
10	10.	1760.2
11	11.	2200.5
12	12.	2200.5

[12] The results of the modal analysis after "tuning" the string. Create mode shape results. ↓



[14] The third mode shape can be deceptive. All numerical values are essentially zeros (the nonzero values are numerical errors). It is simply a straight line. The mode is the rotation in X-direction; it is a rotation mode. We could have fixed X-rotation at both ends to avoid this mode. ↓

[13] We first examine some trivial modes. Highlight **Total Deformation 3**, turn off **Display/Thick Shells and Beams**, and select **Auto Scale** for a better view of the results. ←



[15] The sixth mode is a stretching mode. #

11.4.3 Harmonic Series

[1] A *harmonic mode* has a frequency that is an integral multiplication of the fundamental frequency. The first 5 harmonic modes of the guitar string are shown below [2-6].

If you pluck a string, you will produce a tone made up of all harmonic modes. Although all the plucks produce the same note (in this case, note A), the *harmonic mixes* determine the quality of the note. If you pluck the string near the midpoint of the string, you will produce a tone dominated by the first harmonic [2]. If you pluck the string near the quarter point (in a guitar, that is near the sound hole), you will produce a tone dominated by the second harmonic [3]. And so forth. You can produce an "overtone" (a tone made up of harmonics that are all above the fundamental mode) by touching the string lightly at the midpoint and, at the same time, plucking the string; the first harmonic mode will be suppressed. You can produce other overtones in a similar way.

Different musical instruments generate tones that have different harmonic mixes. A trumpet usually produces much higher harmonics in its frequency spectrum. This gives the trumpet a "brassy" sound. A flute usually produces a tone dominated by the first harmonic with almost no higher harmonics. This gives the flute a unique "pure" sound.

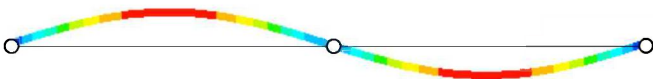
Knowing these physics, an engineer should be able to produce sounds of any musical instruments (including human voices) using a frequency-generating device. ↘

Type: Total Deformation
Frequency: 440. Hz
Unit: mm



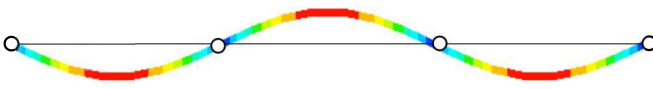
[2] The first harmonic mode (modes 1 or 2) has no nodes between the two ends. ↓

Type: Total Deformation
Frequency: 880. Hz
Unit: mm



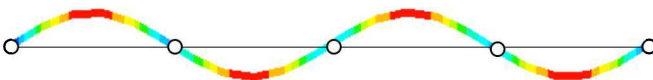
[3] The second harmonic mode (modes 4 or 5) has one node between the two ends. ↓

Type: Total Deformation
Frequency: 1320. Hz
Unit: mm



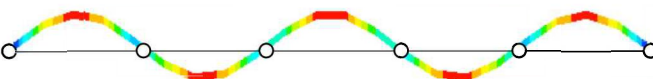
[4] The third harmonic mode (modes 7 or 8) has two nodes between the two ends. ↓

Type: Total Deformation
Frequency: 1760.2 Hz
Unit: mm



[5] The fourth harmonic mode (modes 9 or 10) has three nodes between the two ends. ↓

Type: Total Deformation
Frequency: 2200.5 Hz
Unit: mm



[6] The fifth harmonic mode (modes 11 or 12) has four nodes between the two ends. #

11.4.4 Just Tuning System^[Ref 1]

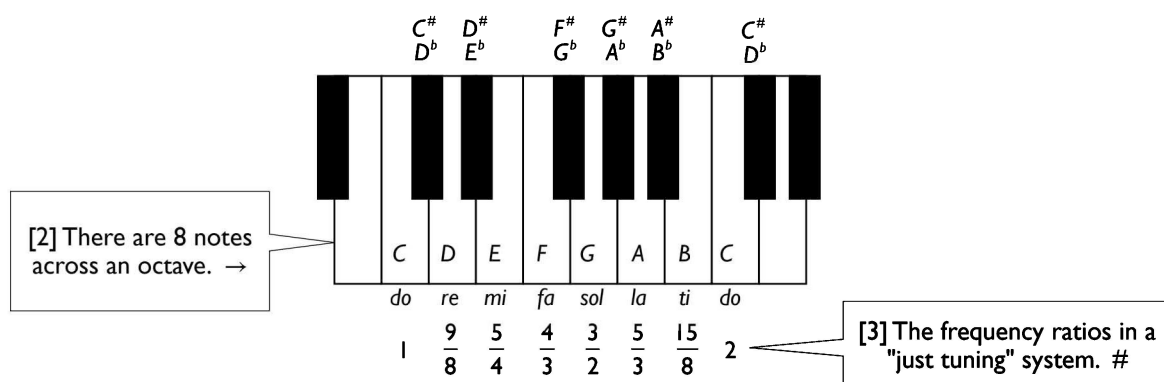
[1] Why do some notes sound pleasing to our ears when played together, while others do not? We know from the experience that when two notes have a simple frequency ratio, they sound harmonious with each other. The simpler the ratio, the more harmonious it sounds; we'll explain this in 11.4.6 (page 416).

In Western music, an 8-tone musical scale has traditionally been used. When learning to sing, we identify the eight tones in the scale by the syllables *do, re, mi, fa, sol, la, ti, do*. For a C-major scale in a piano, there are 8 white keys from a C to the higher pitch of C [2]. The two C's have a frequency ratio of 2:1, and are said to be an octave apart. If we play two notes an octave apart, they sound very similar. In fact, we often have difficulty telling the difference between two notes an octave apart. This is because, except for the fundamental harmonic of the lower note, two notes have most of the same higher harmonics.

For the following discussion, let's arbitrarily assume the frequency of the lower pitch C as 1. (In a modern piano, the middle C has a frequency of 261.63 Hz; see 11.4.5[2], next page.) Then the frequency of the higher pitch C is 2. Before being replaced by the "equal temperament" (11.4.5) in the early 20th century, the "just tuning" systems prevailed in the music world. In a just tuning music system, the frequencies of the notes between the 2 C's are chosen according to the "simple ratio" rule, in order to be harmonious to each other. They are summarized in [3]. Note that we didn't show the frequency ratios for the black keys (the semitones) to simplify our discussion.

Now, you can appreciate that if we play the notes *do* and *sol* together, the sound is pleasing to our ears, since they have the simplest frequency ratio between 1 and 2. You also can appreciate that the major cord C consists of the notes *do, me, sol, do*, the simplest frequency ratios (but not too "close," to avoid *beats*; see 11.4.6, page 416) between 1 and 2.

The problem of the just tuning system is that it is almost impossible to play in another key. For example, when we play in D key, then the frequency ratio between D and its *fifth* (A) is no longer 3/2. Instead, the frequency ratio is an awkward 40/27; the two notes are not harmonious enough any more. ✓

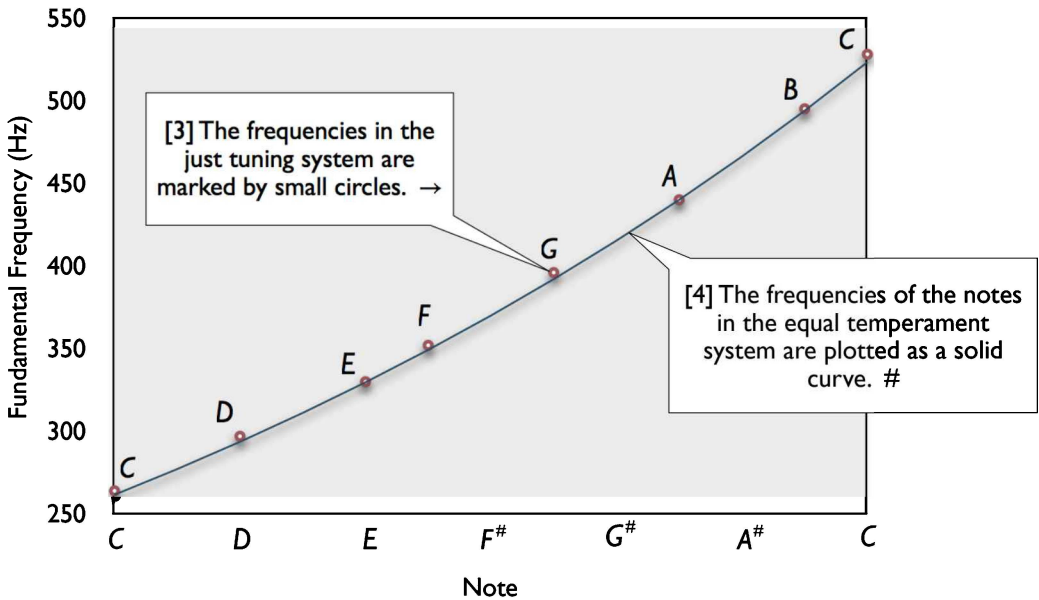


11.4.5 Twelve-Tone Equally Tempered Tuning System^[Ref 2]

[1] Modern Western music is dominated by a 12-tone *equally tempered tuning system*, or simply *equal temperament*. The idea is to compromise the frequency ratios between the notes, so that they can be played in different keys. In this system, an octave is equally divided into 12 tones (including semitones) in logarithmic scale. In other words, the adjacent tones have a frequency ratio of $2^{1/12}$, or 1.05946. For example, the frequency ratio between the C# and the C is $2^{1/12}$; the frequency ratio between an A and the lower C is $2^{9/12}$. According to this idea, frequencies of the notes can be calculated and listed in [2], next page. For comparison, we also list the frequencies of the notes in the just tuning system. The data in the table are plotted into a chart as shown in [3-4]. The compromised frequencies are close enough to the just tuning system that most musicians have been satisfied with this system for centuries. ↵

Note	Just Tuning		Equal Temperament	
	Frequency Ratio	Frequency	Frequency Ratio	Frequency
C	1	264.00	1	261.63
C# (Db)			$2^{1/12}$	277.18
D	9/8	297.00	$2^{2/12}$	293.66
D# (Eb)			$2^{3/12}$	311.13
E	5/4	330.00	$2^{4/12}$	329.63
F	4/3	352.00	$2^{5/12}$	349.23
F# (Gb)			$2^{6/12}$	369.99
G	3/2	396.00	$2^{7/12}$	392.00
G# (Ab)			$2^{8/12}$	415.30
A	5/3	440.00	$2^{9/12}$	440.00
A# (Bb)			$2^{10/12}$	466.16
B	15/8	495.00	$2^{11/12}$	493.88
C	2	528.00	2	523.25

[2] The frequencies of the notes. ✓



11.4.6 Beat Frequency^[Refs 3, 4]

[1] Back to the question in the beginning of 11.4.4[1], page 414: Why do some notes sound pleasing to our ears when played together, while others do not? Why does a simple frequency ratio imply a harmonious sound? To explain this, another physical phenomenon called *beats* is at work.

When two waves of different frequencies are combined, they interfere with each other. When they are in phase, the combined wave has a large amplitude. When they are out of phase, the amplitude becomes smaller. This fluctuation in amplitude of the combined wave is called *beats*, and the frequency is called the *beat frequency*.

The beat frequency is equal to the frequency difference of the two waves. If the two notes are very close in frequency, the beat frequency is slow enough to be heard as a variation in amplitude, that is, you can hear the sound getting louder and softer in a repetitive pattern. This effect can be useful in tuning a musical instrument, since the beats disappear when two frequencies are in tune.

What happens when we play *C* and *D* together? The beat frequency is 32.03 Hz ($293.66 - 261.63$, see 11.4.5[2], last page), which is large enough that we hear a harsh buzz, which is unpleasant for our ears. In music, however, a dissonant sound is sometimes used to produce desired effects, for example, a sad mood.

What happens when we play *C* and *G* together? The beat frequency is 130.37 Hz ($392.00 - 261.63$, see 11.4.5[2], last page), which is very close to one half of the middle *C* (261.63 Hz). In other words, the beat frequency is an octave below middle *C*. Therefore, it fits nicely within a chord containing *C* and *G*. Thus, beats can explain the harmony when we play major chords. ↓

Wrap Up

[2] Save the project and exit Workbench. #

References

1. Wikipedia>Just Intonation.
2. Wikipedia>Equal Temperament.
3. Wikipedia>Beat.
4. Griffith, W.T., The Physics of Everyday Phenomena, Fourth Edition, McGraw-Hill, 2004.
5. <http://plasticity.szynalski.com/tone-generator.htm>

Section 11.5

Review

11.5.1 Keywords

Choose a letter for each keyword, from the list of descriptions

- | | |
|--|--|
| 1. () Dynamic Behavior | 6. () Natural Frequencies |
| 2. () Free Vibration | 7. () Rigid Body Modes |
| 3. () Fundamental Natural Frequency | 8. () Structural Dynamic Analysis |
| 4. () Harmonic Mode | 9. () Transient Structural Analysis |
| 5. () Modal Analysis | |

Answers:

1. (B) 2. (E) 3. (G) 4. (I) 5. (C) 6. (F) 7. (H) 8. (A) 9. (D)

List of Descriptions

- (A) Technique used to determine the dynamic behavior of a structure.
- (B) Includes vibration characteristics, effect of harmonic loads, and effect of general time-varying loads.
- (C) A dynamic analysis to investigate the vibration characteristics, specifically the frequencies and shapes, of a structure without any time-varying loads. The frequencies are called natural frequencies.
- (D) A dynamic analysis to investigate the response of a structure under general time-varying loads.
- (E) The vibration of a structure without any external forces.
- (F) The frequencies corresponding to free vibrations.
- (G) The lowest frequency of free vibration.
- (H) They have infinite period, or, equivalently, zero frequencies. In general, these modes should not be present.
- (I) It has a frequency that is an integer multiple of the fundamental frequency.

11.5.2 Additional Workbench Exercises

Gravity Is Negligible When Evaluating Frequencies of the Two-Story Building

In 11.2.5[2] (page 399), we mentioned that the earth's gravity affects natural frequencies of the two-story building very little. Verify this.

Improving the Stiffness of the Two-Story Building

In 11.2.5[9] (page 401), we mentioned that if we want to further improve the structure's solidity, we may add bracing members in the Z-direction. Implement this idea.

Model Airplane Wing

A wing of a model airplane is detailed in the ANSYS Help System^[Ref 1]. Carry out the modal analysis for the airplane wing. How do you stiffen the wing further without increasing the weight?

Reference

1. ANSYS Help//Mechanical APDL//Mechanical APDL Introductory Tutorials//8. Modal Tutorial

Chapter 12

Transient Structural Simulations

In the real world, all loads are time-varying, so are the structural responses. For example, imagine that you hang a block on a spring and slowly release it. The force on the spring increases gradually from zero until it reaches the weight of the block, and then the block moves up and down for a while, and finally steadies at a certain position, due to the damping of the system. To know the whole process, you need to perform a dynamic simulation. If you are concerned about only the final state (final position), called the *steady state*, of the response, a static simulation is adequate. The response before the steady state is called a *transient state*. A situation where a dynamic simulation can be replaced by a series of static simulations is that if the structure displaces so slowly that the dynamic effects (inertia and damping effects) are negligible. A series of time-varying static simulations is called a *quasistatic simulation*.

Other than these two categories of cases (cases of finding a steady state solution or cases that the structure displaces slowly), dynamic effects must be taken into account and dynamic simulations are needed.

Purpose of This Chapter

The purpose of this chapter is to provide background knowledge as well as practical examples for students to master techniques for dynamics simulations. It is a sequel of the last chapter, modal analysis, in which no external forces are involved. Except Section 12.3, in which **Harmonic Response** analyses are carried out, all the other exercises are performed with **Transient Structural** analysis system, which uses an implicit integration method to calculate the response. **Explicit Dynamics** analysis system, on the other hand, uses an explicit integration method. The explicit dynamics will be introduced in Chapter 15, where differences between implicit and explicit methods will be discussed.

About Each Section

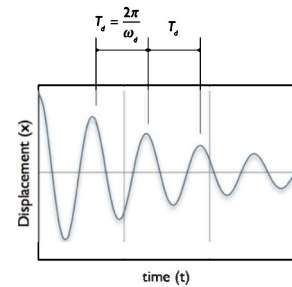
Section 12.1 intends to equip students with background knowledge of structural dynamics. Again, it is the concepts, rather than the mathematics details, that we want to emphasize. We first use a single-degree-of-freedom structural system to illustrate some ideas, and then conceptually generalize these ideas for multiple-degrees-of-freedom structural systems.

Section 12.2 provides a practical example to demonstrate the application of dynamic loads and other considerations such as integration time steps and damping. Section 12.3 uses the two-story building (7.3, 11.2) as an example, demonstrating the procedure of harmonic response analysis. Section 12.4 performs an impact simulation. One of its purposes is to demonstrate how to specify a simple initial condition, namely uniform velocity. Another purpose is to show the limitation of implicit integration methods and the necessity of explicit methods for high-speed impact simulations.

Section 12.5 is a sequel of Section 11.4. As mentioned in 11.4.3 (page 413), if you pluck a string, you will produce a tone made up of all harmonic modes. We will strum different locations on the string to observe the responses. Another purpose of Section 12.5 is to demonstrate how to specify a more general initial condition. Some initial conditions need a static simulation themselves. In this section, we will use the results of a static simulation as an initial condition for a transient simulation.

Section 12.1

Basics of Structural Dynamics



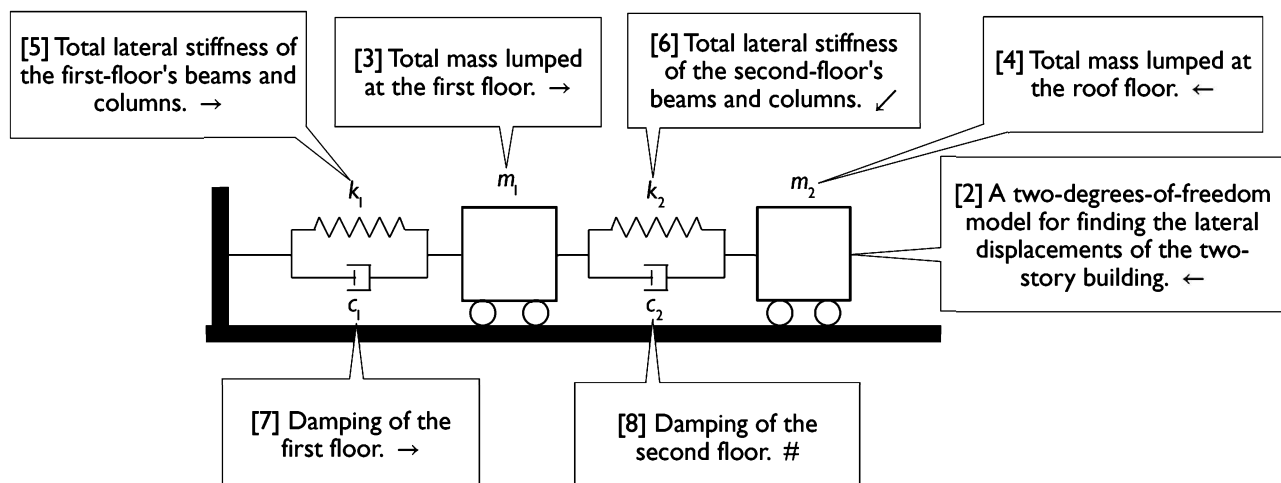
The purpose of this section is to provide basics of structural dynamics, so that students can understand the exercises in this chapter. The concepts and terminology introduced in this section will be used throughout this chapter and the rest of the book. We first use a single-degree-of-freedom lumped mass model to explain some basic behavior of dynamic response. The results will be conceptually extended to multiple-degrees-of-freedom cases (and not limited to lumped mass models).

12.1.1 Lumped Mass Model

[1] In the old days, many dynamics problems were simplified as lumped mass systems of a few degrees of freedom. As an example, to find the lateral displacements, the two-story building (7.3, 11.2) is modeled as a two-degrees-of-freedom system as shown [2-8]. A lumped mass model like this can be solved with a circuit (made of inductors, resistors, capacitors, etc.) that has the same form of governing equations as the lumped mass model. This simple device, used to solve engineering problems long before digital computers became available, is essentially an analog computer.

The parameters ($m_1, m_2, k_1, k_2, c_1, c_2$) must be reasonably evaluated to obtain an acceptable solution. Each of the masses m_1 and m_2 [3-4] may include the floor mass, part of the columns mass, and the equivalent mass of the loads on the floor. Each of the spring constants k_1 and k_2 [5-6] may be calculated according to the bending stiffnesses of the columns and beams at the floor. Each of the damping coefficients c_1 and c_2 [7-8] represents all energy dissipating mechanisms of the floor.

The energy dissipating mechanisms include frictions between the building and the surrounding air (viscous damping), material's internal frictions (material damping), and the frictions in the joints connecting structural members (Coulomb damping). Evaluating damping is one of the most challenging tasks of engineering practice. Fortunately, damping in most real-world structures is usually very small. Consequently, in many cases, it may not be crucial and we may choose a reasonable value according to engineering experiences. In other cases, conducting experiments to evaluate damping values may be needed. We'll discuss damping in 12.1.3, pages 423-425. ↓



12.1.2 Single Degree of Freedom Model

[1] Consider a single-degree-of-freedom (SDOF) model [2]. Applying Newton's law of motion on the block of mass m , we have

$$\sum F = ma$$

$$p - kx - c\dot{x} = m\ddot{x}$$

or

$$m\ddot{x} + c\dot{x} + kx = p \quad (1)$$

Eq. (1) is the governing equation of the SDOF model. We'd assumed that the spring force is linearly proportional to the displacement, and the damping force is linearly proportional to the velocity. This kind of damping is called a *viscous damping*. We will discuss this assumption of damping in 12.1.3, pages 423-425.

If no external forces exist, Eq. (1) becomes

$$m\ddot{x} + c\dot{x} + kx = 0 \quad (2)$$

Eq. (2) represents a *free vibration* system.

If the damping is negligible, then the equation becomes

$$m\ddot{x} + kx = 0 \quad (3)$$

Eq. (3) represents an *undamped free vibration* system.

Undamped Free Vibration

The general solution of Eq. (3) is

$$x = A \sin(\omega t + B) \quad (4)$$

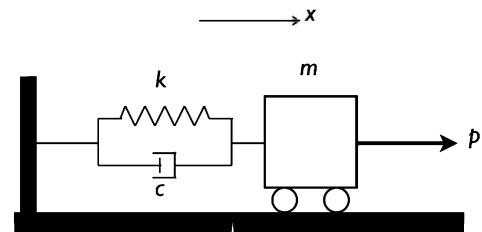
where A and B are arbitrary real numbers and

$$\omega = \sqrt{\frac{k}{m}} \quad (5)$$

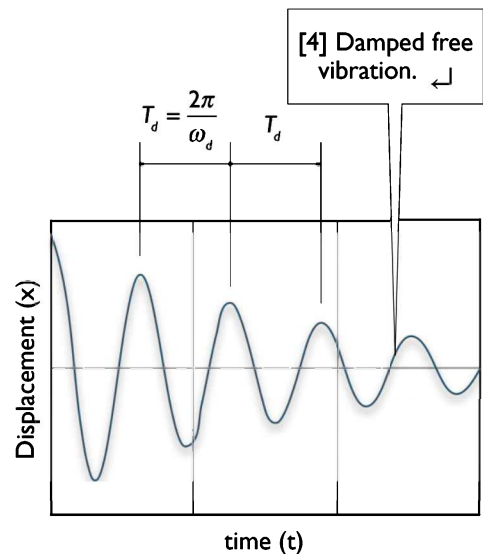
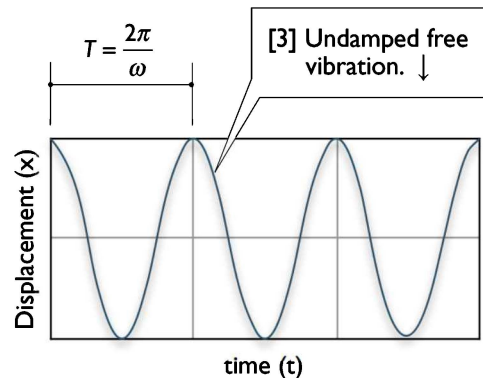
Verification of this solution can be done by substituting Eqs. (4-5) into Eq. (3). A typical plot of Eq. (4) is shown in [3], where we arbitrarily assume $B = \pi/2$, since it is irrelevant. The vibration can be generated by holding up the mass of an undamped SDOF system [2], displacing an arbitrary amount A , releasing it, and letting it vibrate freely. The meaning of the *natural angular frequency* ω is also shown in the plot [3]. Relations between the natural angular frequency ω (rad/s), *natural frequency* f (Hz), and *natural period* T (s) are

$$f = \frac{\omega}{2\pi} \quad (6)$$

$$T = \frac{1}{f} \quad (7) \nearrow$$



[2] A single-degree-of-freedom model. ↓



Damped Free Vibration

[5] Assume the damping c is smaller than the critical damping, defined in Eq. (11), then the general solution of Eq. (2) is

$$x = Ae^{-\xi\omega t} \sin(\omega_d t + B) \quad (8)$$

where A and B are arbitrary real numbers and

$$\omega_d = \omega\sqrt{1-\xi^2} \quad (9)$$

$$\xi = \frac{c}{c_c} \quad (10)$$

$$c_c = 2m\omega \quad (11)$$

Verification of this solution can be done by substituting Eqs. (8-11) into Eq. (2). A typical plot of Eq. (8) is shown in [4] (last page). Again, this vibration can be generated by holding up the mass of a SDOF system [2], displacing an arbitrary amount A , releasing it, and letting it vibrate freely.

The quantity c_c , defined in Eq. (11), is called the *critical damping coefficient*. It can be shown that if the damping c is larger than or equal to c_c , then the Eq. (8) is no longer valid. Actually, the motion is no longer oscillatory, and the system is called *over-damped* (if $c > c_c$) or *critically damped* (if $c = c_c$). For most structures, we may reasonably assume that the system is under-damped ($c < c_c$), and the Eq. (8) is always valid.

The quantity ξ , defined in Eq. (10), is called the *damping ratio*. Values of damping ratio for typical structures range from about 0.02 (e.g., piping systems) to about 0.07 (e.g., bolted structures, reinforced concrete)^[Ref 1].

The quantity ω_d , defined in Eq. (9), is called *damped natural angular frequency*. Note that, for a small damping ratio, ω_d and ω are practically the same. For example, with $\xi = 0.07$, $\omega_d = 0.9975\omega$.

Measuring Damping Coefficient

Given the mass m and the spring constant k of a SDOF system, and if we can obtain a damped free vibration curve [4] from an experiment, then we can calculate the damping coefficient from this information, as follows. From Eq. (8), the peak displacement is

$$x_{\text{peak}} = Ae^{-\xi\omega t}$$

Recognizing the time between two peaks is $T = 2\pi/\omega_d \approx 2\pi/\omega$, we can write down the displacement ratio R between two consecutive peaks

$$R = \frac{x_{\text{peak}2}}{x_{\text{peak}1}} = \frac{Ae^{-\xi\omega(t+\frac{2\pi}{\omega})}}{Ae^{-\xi\omega t}} = e^{-2\pi\xi} \quad (12)$$

For example, with $\xi = 0.07$, the displacement ratio between two consecutive peaks is $R = 0.64$. In practice, the displacement ratio can be calculated by averaging the displacement ratios of several cycles of vibrations. The damping ratio ξ then can be calculated from Eq. (12)

$$\xi = \frac{-\ln R}{2\pi} \quad (13)$$

The damping coefficient, if desired, can be calculated using Eqs. (10) and (11),

$$c = \frac{-\ln R}{\pi} m\omega \quad (14) \quad \leftarrow$$

Viscous damping coefficient c is not an intrinsic property of a material

[6] Imagine that the SDOF system [2] represents a cantilever beam made of a material and you want to characterize the damping property for the material. You gather a specimen, generate a free vibration curve [4], and calculate the damping coefficient according to Eq. (14). It seems easy. The problem is that, in this way, the damping coefficient depends on the geometry of the specimen: different geometries have different damping coefficients. To characterize damping for a material, we need more knowledge of damping mechanisms. #

12.1.3 Damping

Damping Mechanisms

[1] As mentioned, damping includes all energy dissipating mechanisms. In a structural system, all energy dissipating mechanisms boil down to one word: friction. In a structure, three categories of frictions can be identified: First, friction between the structure and its surrounding fluid, called *viscous damping*. Second, internal friction in the material, called *material damping*, *solid damping*, or *elastic hysteresis*. Third, friction in the connection between structural members, called *dry friction* or *Coulomb friction*. When the structure is surrounded by the air, the viscous damping is usually very small, and the major sources of damping are material damping and Coulomb friction.

Viscous Damping

Viscous damping is the friction between a structure and its surrounding fluid. If small, viscous damping force can reasonably assume to be proportional to the velocity of the structural displacement

$$F_D = c\dot{x} \quad (1)$$

The viscous damping coefficient c can be input directly as an element parameter, such as a spring element [2]. For each material, you can include a constant damping ratio as a material property (see [13], page 425). In some analysis systems, the viscous damping can be specified using a global damping ratio ξ [3]. The damping coefficient c is then calculated according to Eqs. 12.1.2(10-11) (last page),

$$c = 2m\omega\xi = 2\xi\sqrt{mk} \quad (2)$$

In general, to introduce viscous damping to a structure, you may add individual elements involving viscous damping, such as spring elements [2]. In **Harmonic Response** analysis (Section 12.3), which calculates the response under various frequencies, viscous damping can be specified using a global damping ratio [3]. ✓

Details of "Longitudinal - Ground To Solid"	
+ Graphics Properties	
[-] Definition	
Type	Longitudinal
Spring Behavior	Both (Linear)
Longitudinal Stiffness	0. N/m
Longitudinal Damping	2000. Ns/m

[2] For spring elements, viscous damping coefficient can be input as an element parameter. →

Details of "Analysis Settings"	
+ Options	
+ Output Controls	
[-] Damping Controls	
Constant Damping Ratio	2.e-002
Stiffness Coefficient Define By	Direct Input
Stiffness Coefficient	0.
Mass Coefficient	0.
+ Analysis Data Management	

[3] In Harmonic Response analysis, viscous damping can be specified using a global damping ratio. ↵

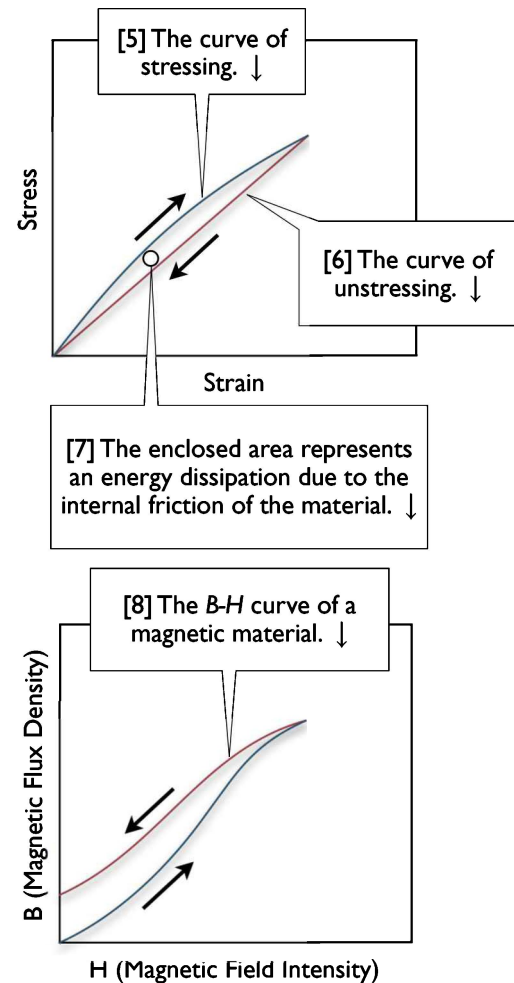
Material Damping

[4] Material damping is the internal friction between molecules or grains of the material. Other names for material damping include *solid damping* and *elastic hysteresis*. It is often the major sources of damping in a real-world structure.

To understand the material damping, consider a typical stress-strain relation in a uniaxial material test [5-7]. Unlike the tests shown in 1.4.1 [2-6] (page 40), here we repeatedly increase the stress [5] and then release the stress [6]. If the plastic deformation (permanent deformation) is not present, the strain will return to the original state when releasing the stress, and the curve of releasing stress is likely to be a straight line. A material in which the strain state returns to the original state when the stress is released is called an *elastic material*.

The area enclosed by the curves [7] represents the energy dissipation (to the environment) due to the internal friction of the material. This area is typically very small; the plot [5-7] is exaggerated for instructional purposes.

The plot [5-7] is analogous to a B - H curve of a magnetic material [8], where B is the *magnetic flux density* and H is the *magnetic field intensity*. A magnetic field H is applied on a magnetic material to create a B field within the material (i.e., the material is magnetized). The area enclosed by the B - H curve represents an energy dissipation, and is called the *magnetic hysteresis*. Because of the analogy, the energy dissipation in a stress-strain curve is called the *elastic hysteresis*. ↗



[9] Recognizing that the damping is small for a structure and the global behavior is similar regardless of the sources of damping, Workbench assumes, as a simplification, that the material damping force is proportional to the structural velocity (this somehow deviates from reality, in which the material damping force is more likely proportional to the structural displacement rather than the structural velocity), the same as the viscous damping,

$$F_D = c\dot{x} \quad (3)$$

However, we cannot characterize a material using a damping coefficient c , since, as mentioned in 12.1.2[6] (page 423), the damping coefficient c is not an intrinsic property of a material. To filter out factor of geometry, we need more elaboration. Eq. (2) shows how the coefficient c relates to the mass m and stiffness k for the case of single degree of freedom; for cases of multiple degrees of freedom, the relation is not so simple. In engineering practice, an efficient way to characterize a material is proposing a mathematics form with parameters and then determining the parameters using data fitting. With this idea, the coefficient c is assumed a linear combination of the mass m and the stiffness k [Ref 1],

$$c = \alpha m + \beta k \quad (4)$$

Now, the parameters α and β are used to characterize the damping property of a material. Eq. (4) is based on an observation that damping is related to the mass and the stiffness of the structure. ↵

[10] Using $c = 2m\omega\xi$ (Eq. (2)) and $k = m\omega^2$ (Eq. 12.1.2(5), page 421), we may rewrite Eq. (4) in terms of frequency and damping ratio,

$$2\omega\xi = \alpha + \beta\omega^2 \quad (5)$$

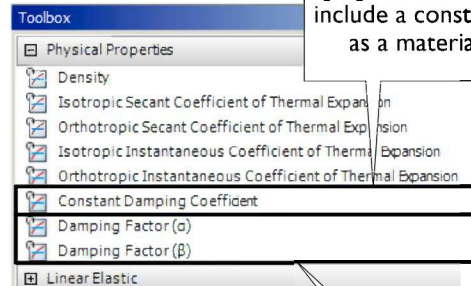
If we can make a single material specimen and measure the damping ratios ξ_i under different excitation frequencies ω_i , or make several material specimens of different sizes, and measure the damping ratios ξ_i under their respective fundamental frequencies ω_i , or, even better, a combination of the above ideas, then we can evaluate the material parameters α and β by a standard data fitting procedure.

Workbench allows you to input the α and β values for each material as material properties [11]. A **Transient Structural** analysis system also allows you to input a global β value (**Stiffness Coefficient**) and a global α value (**Mass Coefficient**), in a details view of **Analysis Settings** [12]. ↓

[12] A global beta value (**Stiffness Coefficient**) and alpha (**Mass Coefficient**) can be input in the **Damping Controls** section of a details view of **Analysis Settings**. →

Damping Controls	
Stiffness Coefficient Define By	Direct Input
Stiffness Coefficient	2.e-003
Mass Coefficient	0.
Numerical Damping	Program Controlled
Numerical Damping Value	0.1

[13] For each material, you can include a constant damping ratio as a material property. ✓



[11] Alpha and beta values as material properties can be included. ↖

[14] If we assume $\alpha = 0$, Eq. (5) becomes

$$\beta = \frac{2\xi}{\omega} \quad (6)$$

Eq. (6) is a simple relation between the β value and the damping ratio ξ . It can be used to estimate one value when knowing the other one, if the frequency is also known.

Although the damping ratio is meant to be used for the viscous damping ([3], page 423) rather than the material damping, in practice, the engineers often use a damping ratio to simplify the overall damping effect, when the damping is not critical to the response. As mentioned in 12.1.2[5] (page 422), values of damping ratio for typical structures range from 0.02 to 0.07.

Coulomb Friction

Another major source of damping is the friction in the connection between structural members. It is called the *dry friction* or *Coulomb friction*. In Workbench, it is implemented as *frictional contacts*. To include the Coulomb friction, you have to specify frictional contacts between parts, which will be discussed in Chapter 13. #

12.1.4 Analysis Systems

[1] Generally, we are dealing with multiple-degrees-of-freedom systems. The foregoing concepts may be extended for general cases. Specifically, Eq. 12.1.2(1) (page 421) can be generalized for multiple-degrees-of-freedom cases,

$$[M]\{\ddot{D}\} + [C]\{\dot{D}\} + [K]\{D\} = \{F\} \quad (1)$$

Where $\{D\}$ is the nodal displacements vector, $\{F\}$ is the nodal external forces vector, $[M]$ is the *mass matrix*, $[C]$ is the *damping matrix*, and $[K]$ is the *stiffness matrix*. Eq. (1) also can be viewed as a generalization of Eq. 1.3.1(1) (page 35).

Eq. (1) represents the governing equation of a transient structural simulation. It can be viewed as a force equilibrium relation. On the right hand side of the equation is the external force $\{F\}$. On the left hand side, the first item $[M]\{\ddot{D}\}$ is the *inertia force*, the second item $[C]\{\dot{D}\}$ is the *damping force*, and the third item $[K]\{D\}$ is the *elastic force*. Combination of the inertia force, damping force, and the elastic force balances with the external force.

Let's look at some specialized cases of Eq. (1) (also see [2]). ↘

Modal Analysis

[3] Imagine that you displace a structure a certain amount and then release. There is no external force involved; it is called a free vibration. Eq. (1), since there is no external force, becomes

$$[M]\{\ddot{D}\} + [C]\{\dot{D}\} + [K]\{D\} = 0 \quad (2)$$

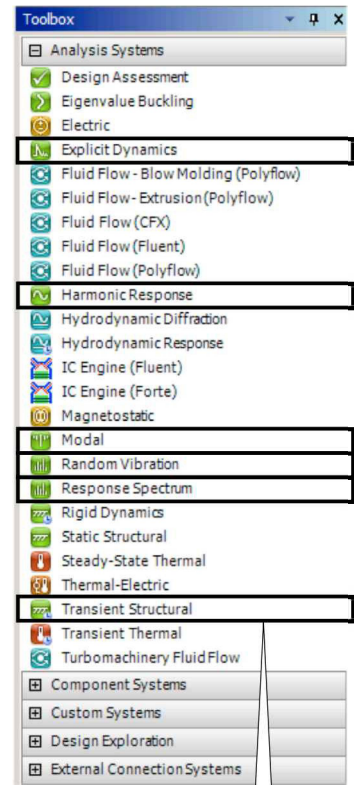
Solution of Eq. (2) is not unique. For a problem of n degrees of freedom, it has at most n solutions, denoted by $\{D_i\}, i = 1, 2, \dots, n$. These solutions are called *mode shapes* of the structure. Each mode shape $\{D_i\}$ can be resonantly excited by an external excitation of frequency ω_i , called the *natural frequency* of the mode. The lowest frequency is called the *fundamental natural frequency*, or simply *fundamental frequency*. Finding all or some of the mode shapes and their corresponding natural frequencies is called a *modal analysis*.

In a modal analysis, since we are usually interested only in the natural frequencies and the relative shapes of the vibration modes (the absolute values of deformation depend on the energy that excites the structure), the damping effect is usually neglected (see Eq. 12.1.2(9), page 422) to simplify the calculation; Eq. (2) becomes

$$[M]\{\ddot{D}\} + [K]\{D\} = 0 \quad (3)$$

It is Eq. (3) that Workbench solves in a **Modal** analysis system. Note that modal analysis is a linear analysis; all nonlinearities are ignored.

We've performed some modal analyses in Chapter 11. ↓



[2] Structural dynamic simulations can be classified into various analysis types. ←

Transient Structural Analysis

[4] **Transient Structural** analysis solves the general form of Eq. (1). External force $\{F\}$ can be time-dependent forces. All nonlinearities can be included. It uses a *direct integration method* to calculate the dynamic response. In this chapter, all exercises, except Section 12.3, are carried out with **Transient Structural** system [2]. Through these exercises, we will demonstrate how to specify initial conditions, dynamic loads, and set up **Analysis Settings**. ↵

Harmonic Response Analysis

[5] Imagine that a rotatory machine is installed on the floor of the two-story building (Sections 7.3, 11.2), its operational speed 3000 rpm. Due to an inevitable eccentricity of the rotation, the machine generates an up-and-down harmonic force on the floor. After it is started up, the machine's speed increases from zero up to 3000 rpm. Any natural frequencies of the building ranging from zero to 3000 rpm may be excited by the harmonic force. While vibrations of the building or its structural components are unavoidable, the question is how large the amplitude of the vibrations will be. Do the vibrations cause any safety concern or psychological annoyance? A **Harmonic Response** analysis can be carried out to answer that question.

Harmonic Response analysis solves a special form of Eq. (1), in which the external force on i th degree of freedom is of the form

$$F_i = A_i \sin(\Omega t + \phi_i) \quad (4)$$

where A_i is the amplitude of the force, ϕ_i is the phase angle of the force, and Ω is the angular frequency of the external force. Due to the special form of the external forces, the calculation is much more efficient than a general transient response analysis. The steady-state solution of the equation will be of the form

$$D_i = B_i \sin(\Omega t + \varphi_i) \quad (5)$$

The goal of the harmonic response analysis is to find the magnitude B_i and the phase angle φ_i of the response for each degree of freedom, under a range of frequencies of the external forces. Note that harmonic response analysis is a linear analysis; all nonlinearities are ignored.

In Section 12.3, we will use the two-story building to demonstrate the procedure of harmonic response analysis.

Explicit Dynamics

Similar to **Transient Structural**, **Explicit Dynamics** also solves the general form of Eq. (1). The external force $\{F\}$ can be time-dependent forces. All nonlinearities can be included. It also uses a direct integration method to calculate the dynamic response. The difference is described as follows.

The direct integration method used in **Transient Structural** analysis is called an *implicit integration method*. The implicit method works fine for most applications except for high-speed impact simulations.

In high-speed impact simulations, the duration of impact time is so short that the integration time needs to be extremely small (e.g., micro to nano seconds) to catch the details of the behavior. If the implicit integration method is used, the total number of time steps becomes so huge that the computational time is unbearable. That calls for an *explicit integration method*, implemented in **Explicit Dynamics** analysis system.

For many transient dynamic simulations, the explicit method is not popular for one reason: it requires very small integration time steps to achieve an accurate solution. A small integration time is exactly what a high-speed impact simulation needs; therefore, it is not a disadvantage any more. The advantage, on the other hand, is that the calculation is very efficient in each time step. Overall, the high-speed impact simulations are possible only with the explicit integration method.

There are cases, other than high-speed impact simulations, that benefit by using the explicit method. Highly nonlinear simulations usually require very small time steps to overcome the convergence difficulties. In such cases, explicit method may be used.

In this chapter, we will restrict the discussion to implicit dynamics only (i.e., using **Transient Structural** analysis system). The applications of **Explicit Dynamics** will be postponed until Chapter 15. The reason is that, since explicit dynamics usually involves nonlinearities, we need more background on nonlinear simulations, which will be covered in Chapters 13 and 14.

For the beginners, the origination of the names "implicit" or "explicit" may not be important. You may regard them as code names. For those students with strong curiosity, you will learn them in Section 15.1. ↩

Response Spectrum Analysis

[6] We often design an engineering object such that it can withstand oscillatory or repeated loadings. For example, a building must withstand the strikes of earthquakes. When designing a building, we may use a well-recorded earthquake as a "design earthquake." The history of the earthquake (typically a history of acceleration versus time) can be input as loads and a transient structural simulation is then carried out. After the simulation, the maximum stress (or any other responses) at each location of the structure is collected. The members of the structure are designed according to these maximum values of responses. If the maximum response is the sole purpose of the simulation, a much more inexpensive way of simulation is available: response spectrum analysis.

A time history of earthquake can be transformed to a response spectrum, a maximum response versus frequency (or period) plot. The response spectrum is then input to a **Response Spectrum** analysis system. The output is the maximum response (e.g., maximum stress) at each location of the structure.

Random Vibration Analysis

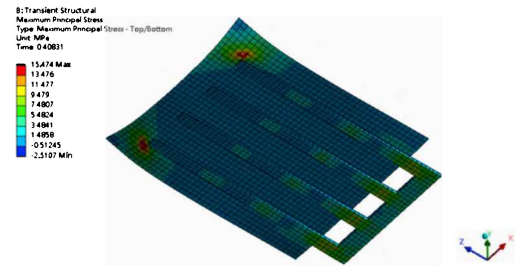
There exists a design methodology that considers probabilistic loads, instead of deterministic loads. The probabilistic loads are described by a spectrum representing probability distribution of excitation at varying frequencies in known directions. The technique is used to design structures withstanding probabilistic loadings, for example, a space vehicle subject to probabilistic strikes of meteorites or asteroids. #

References

1. Cook, R.D., Milkus, D. S., Plesha, M. E., and Witt, R. J., *Concepts and Applications of Finite Element Analysis, Fourth Edition*, John Wiley & Sons, Inc., 2002; Section 11.5 Damping.
2. All Help>Mechanical APDL>Theory Reference>14.3. Damping Matrices

Section 12.2

Lifting Fork

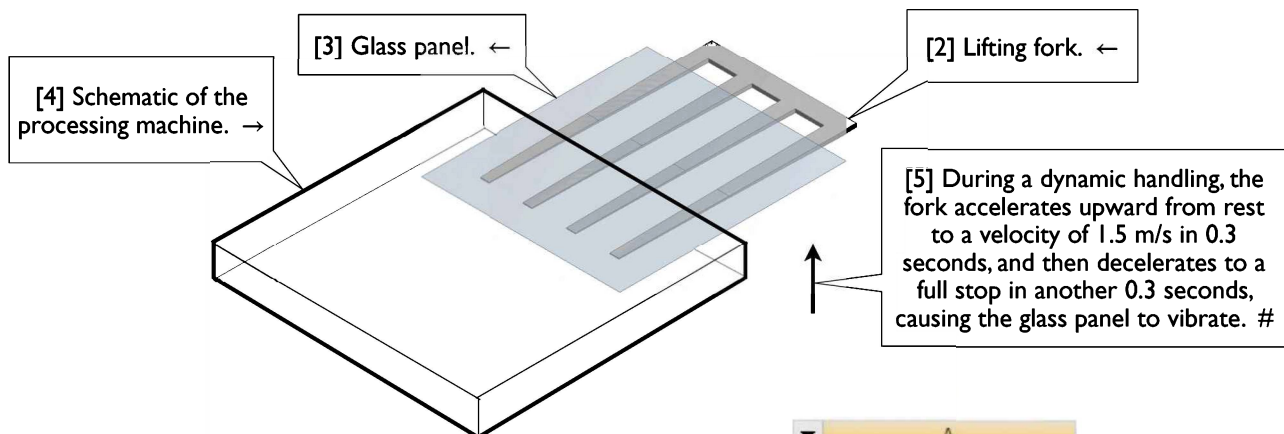


12.2.1 About the Lifting Fork

[1] In Section 4.3, we built a model for a lifting fork and glass, in which the fork was modeled as solid body and the glass as surface body. The lifting fork [2] is used in an LCD factory to handle a glass panel [3], which is so large and thin that the engineers are concerned about its vertical deflections during dynamic handling.

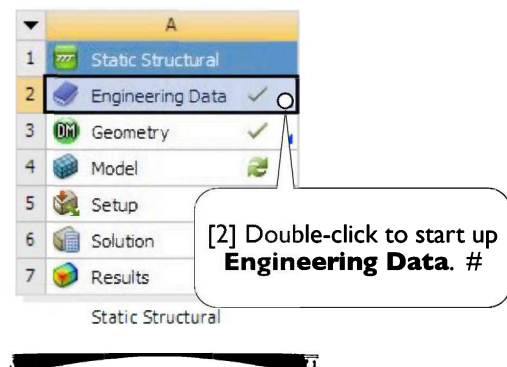
The fork is made of steel with a density of 7850 kg/m^3 , Young's modulus of 200 GPa , and Poisson's ratio of 0.3 . The glass has a density of 2370 kg/m^3 , Young's modulus of 70 GPa , and Poisson's ratio of 0.22 .

In this section we will perform a static structural simulation first, to evaluate the vertical deflection of the glass panel under the gravitational force. This is critical when determining the clearance of the processing machine [4]. During a dynamic handling, the fork accelerates upward from rest to a velocity of 1.5 m/s in 0.3 seconds, and then decelerates to a full stop in another 0.3 seconds, causing the glass panel to vibrate [5]. We want to know the time duration when the vibration is settled to a certain amount so that the glass can be moved into the processing machine [4]. We also want to know the maximum stress during the handling. ↘



12.2.2 Resume the Project Fork

[1] Launch Workbench. Open the project **Fork**, which was saved in Section 4.3. →



12.2.3 Set Up the Model

[1] Click to add a new material and type the name **Glass**.

[2] Double-click to include **Density**.

[3] Double-click to include **Isotropic Elasticity**.

[4] Type the properties of the glass. Note that the SI unit system is used.

[5] Return to **Project Schematic**.

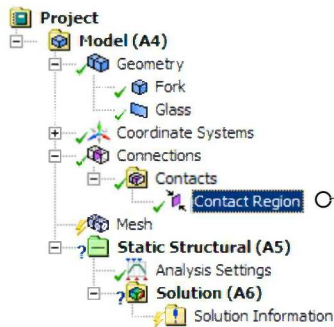
	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	2370	kg m ⁻³		
4	Isotropic Elasticity				
5	Derive from	Young's Modulus and Poisson's Ratio			
6	Young's Modulus	7E+10	Pa		
7	Poisson's Ratio	0.22			
8	Bulk Modulus	4.1667E+10	Pa		
9	Shear Modulus	2.8689E+10	Pa		

[6] Double-click to start up **Mechanical**. Select the **mm-kN-s** unit system.

[7] Highlight **Glass**.

[8] Select **Glass**.

Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Thickness	1 mm
Thickness Mode	Refresh on Update
Offset Type	Middle
Behavior	None
Material	
Assignment	Glass
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	
Statistics	
CAD Attributes	
DMSheetThickness	0.001



Details of "Contact Region"	
Scope	
Scoping Method	Geometry Selection
Contact	5 Faces
Target	1 Face
Contact Bodies	Fork
Target Bodies	Glass
Target Shell Face	Program Controlled
Shell Thickness Effect	No
Definition	
Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Tam Contact	Program Controlled
Tam Tolerance	9.1007 mm
Suppressed	No
Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Pinball Region	Program Controlled

About the Contact Region

[10] Note that, in this case, we use **Bonded** (default) rather than more realistic contact types such as **Frictionless** or **Frictional**, because **Bonded** should be accurate enough for this case; it avoids introducing nonlinearity into the simulation system.

The contact types **Bonded** and **No Separation** are the only two contact types that do not introduce nonlinearity. **No Separation** contact condition between two surfaces prohibits separation in their normal direction, but allows sliding relative to each other. The sliding is assumed to be very small such that the small-deformation theory can apply and the simulation remains linear. ✓

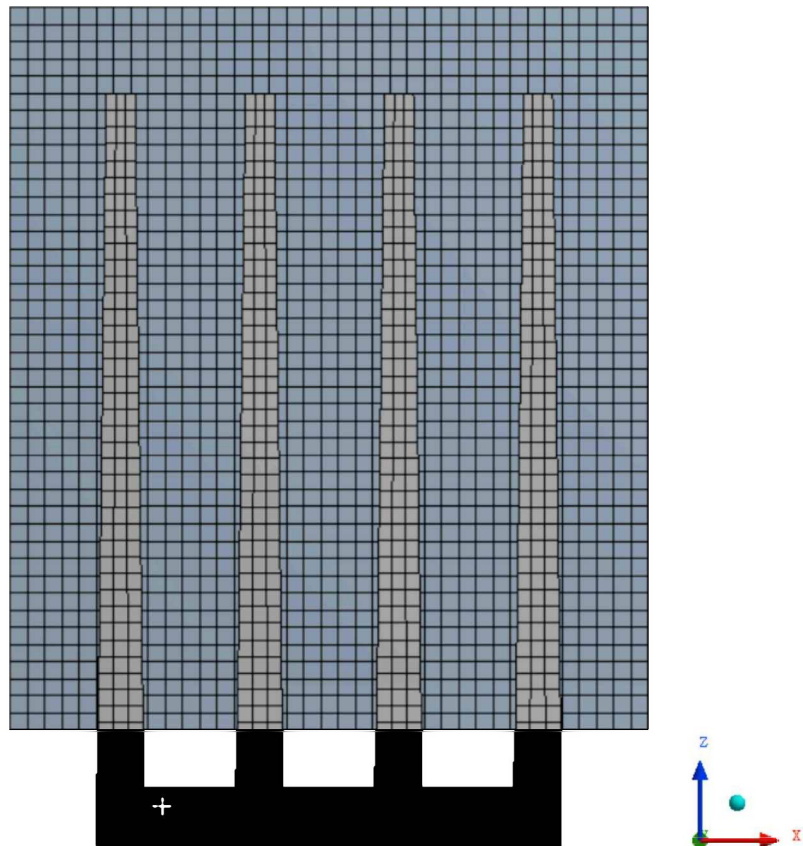
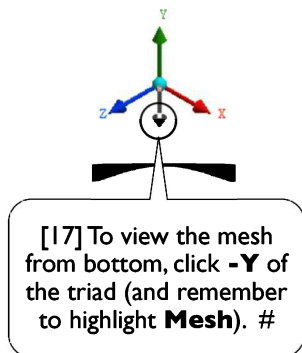
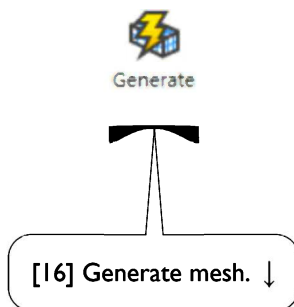
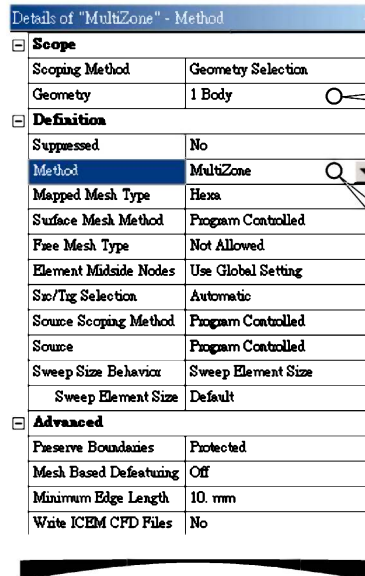
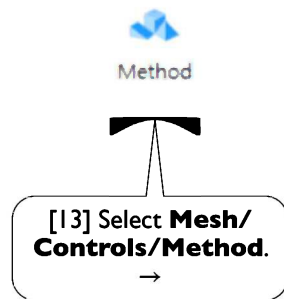


Face
Meshing

[11] With **Mesh** highlighted, select **Mesh/Controls/Face Meshing**. →

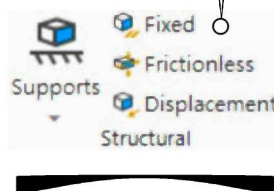
Details of "Face Meshing" - Mapped Face Meshing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Mapped Mesh	Yes
Method	Quadrilaterals
Constrain Boundary	No
Advanced	
Specified Sides	No Selection
Specified Corners	No Selection
Specified Ends	No Selection

[12] Select a face of the glass panel (either top or bottom). **Quadrilaterals** is the default method. ↩

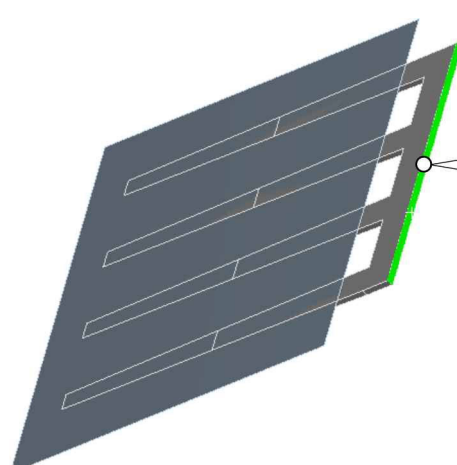


12.2.4 Evaluate Deflection under Gravity

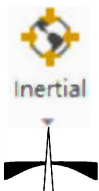
[1] With **Static Structural** highlighted, insert a **Fixed Support**. →



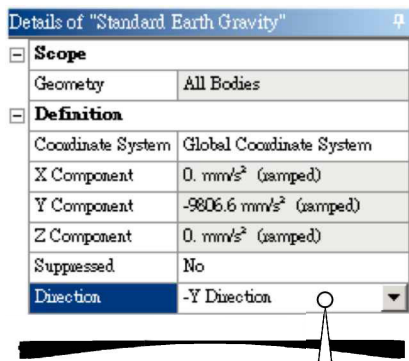
[2] Set up a **Fixed Support** on this face. ✓



[3] Insert an **Inertial/Standard Earth Gravity**. →



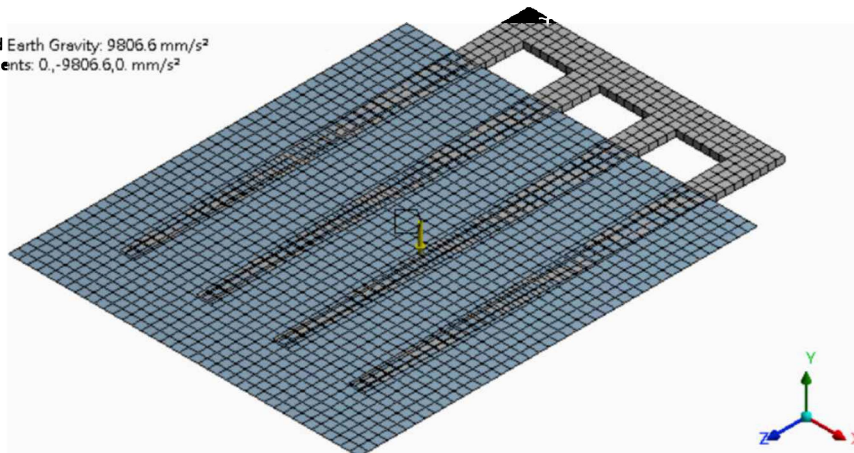
[4] Select **-Y Direction**. This is the direction of gravitational force. ↵

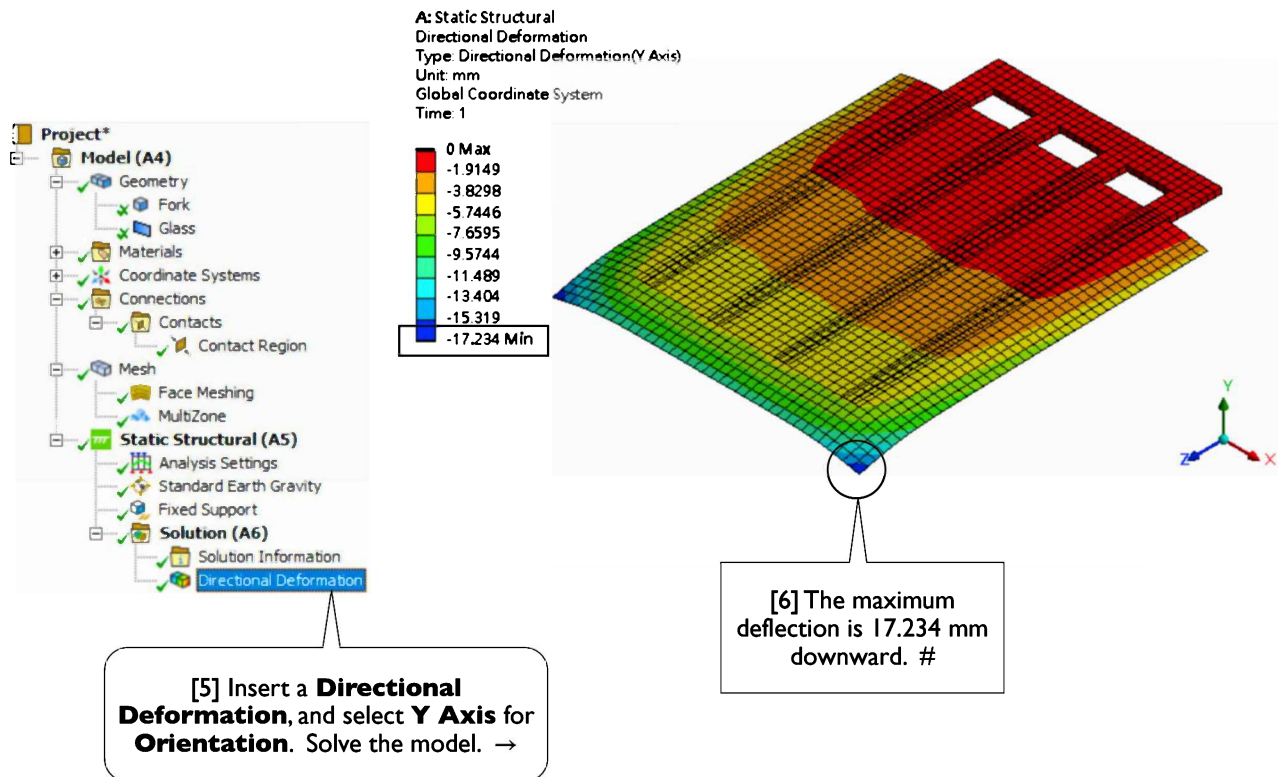


Details of "Standard Earth Gravity"	
Scope	
Geometry	All Bodies
Definition	
Coordinate System	Global Coordinate System
X Component	0. mm/s ² (ramped)
Y Component	-9806.6 mm/s ² (ramped)
Z Component	0. mm/s ² (ramped)
Suppressed	No
Direction	-Y Direction

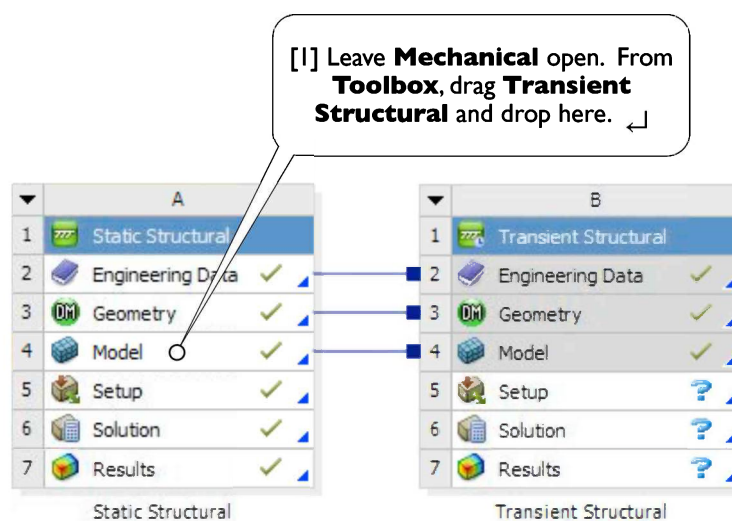
A: Static Structural
Standard Earth Gravity
Time: 1. s

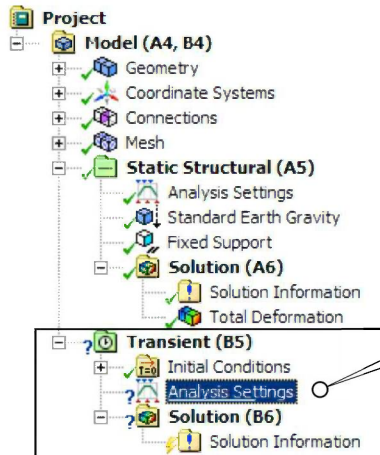
Standard Earth Gravity: 9806.6 mm/s²
Components: 0,-9806.6,0. mm/s²



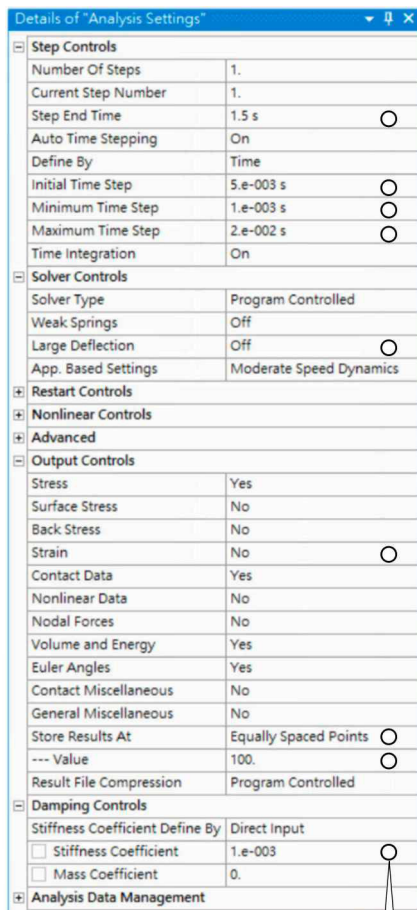


12.2.5 Perform Transient Structural Simulation





[2] Return to **Mechanical**. The project tree is updated to include a **Transient** environment. Highlight its **Analysis Settings**. ✓



[3] Set up **Analysis Settings** like this. →

Step End Time

[4] This is the total simulation time. According to engineers' experience, the vibration should settle to a negligible amount in 1.5 sec.

Initial Time Step

As a guideline, it is suggested that the integration time step be about 1/20 of the response period, to catch the detail behavior of the structural response. The response frequency is estimated to be 10 Hz (a simulation shows that the response frequency is about 6-10 Hz; see [19], page 437). According to this guideline, the integration time step is about

$$ITS = \frac{1}{20f} = \frac{1}{20(10)} = 0.005 \text{ sec}$$

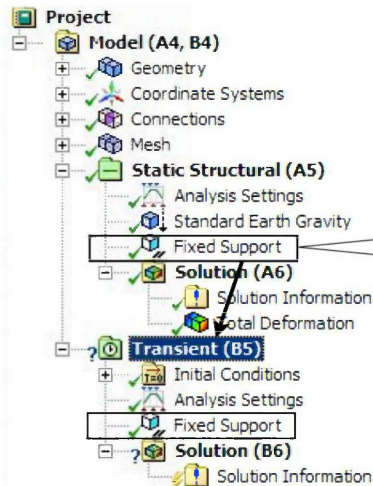
Output Controls

Transient dynamic simulations usually generate huge amounts of data. **Output Controls** allows users to cut down data storage space and computing time.

Beta Damping Value

The beta damping value (**Stiffness Coefficient** in [3]) is reported to be 0.001 from a lab test (see Eq. 12.1.3(4), page 424).

We could specify this value as a material property. Since the vibration is dominated by the glass, here, we choose to input the beta value as a global damping value, that is, neglecting the difference caused by the steel material. ↩

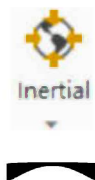


[5] Drag **Fixed Support** from **Static Structural** and drop to **Transient**. Environment conditions can be duplicated this way. →

Why Fixed Support?

[6] During the handling process, the fork is not fixed. It moves in the vertical direction. We could have set up a **Frictionless** support for the same face and allowed the fork to move vertically. Obtaining absolute values of vertical displacement, however, is not what we are concerned about. Our goal is to investigate the vibration behavior. If we let the fork move with such a large displacement, the small displacement of vibrations would be numerically overwhelmed, and you would not see the vibrations at all.

By setting up **Fixed Support** at the face of the fork, and applying inertia force downward (since the fork accelerates upward), we will have the same effect and avoid the numerical problems. ←



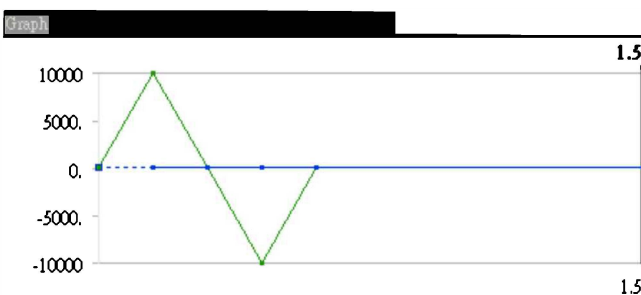
[7] With **Transient** highlighted, select **Inertial/Acceleration**. →

Details of "Acceleration"

Scope	
Geometry	All Bodies
Definition	
Base Excitation	No
Define By	Components
Coordinate System	Global Coordinate System
X Component	Tabular Data
Y Component	Tabular Data
Z Component	Tabular Data
Suppressed	No

[8] Select **Components**. ↓

[10] After the typing, the details look like this. ↓



Steps	Time [s]	X [mm/s²]	Y [mm/s²]	Z [mm/s²]
1	0.	0.	0.	0.
2	0.15	0.	10000	0.
3	0.3	0.	0.	0.
4	0.45	0.	-10000	0.
5	0.6	0.	0.	0.
6	1.5	0.	0.	0.
*				

[9] Type tabular data like this. Type time values on the bottom row (which begins with a *); the time values will automatically insert to appropriate positions. ↑

How are the acceleration data calculated?

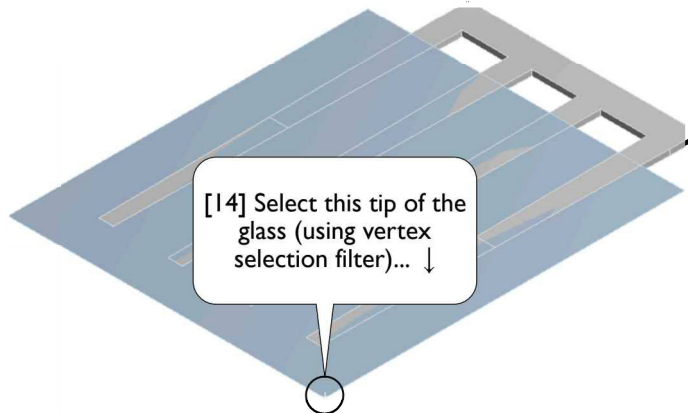
[11] During the handling, the fork accelerates upward to a speed of 1.5 m/s in 0.3 sec and then decelerates to a full stop in another 0.3 sec. The average acceleration is thus 5 m/s^2 . In order to reduce the amplitude of vibrations, the system controls the acceleration such that it increases linearly to 10 m/s^2 (note that this is about the same as a gravitational acceleration) and then decreases to zero linearly. The same idea applies during the deceleration. ↵



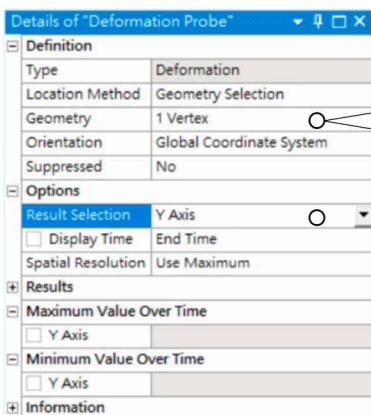
[12] With **Solution of Transient** highlighted, select **Deformation/Directional**, and select **Y Axis** as **Orientation**. →



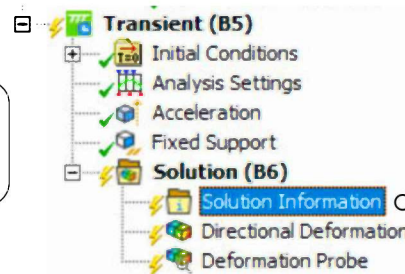
[13] Select **Probe/Deformation**. ✓



[14] Select this tip of the glass (using vertex selection filter)... ↓



[15] And click **Apply** and select **Y Axis**. →



[16] Highlight **Solution Information of Transient**. ✓



[17] Solve. →

[18] Scroll down to examine the text output. It takes many substeps to complete the simulation. Note that this is a linear simulation; there are no nonlinearities involved; each substep needs exactly one iteration. ✓

[19] Response frequency. #

```

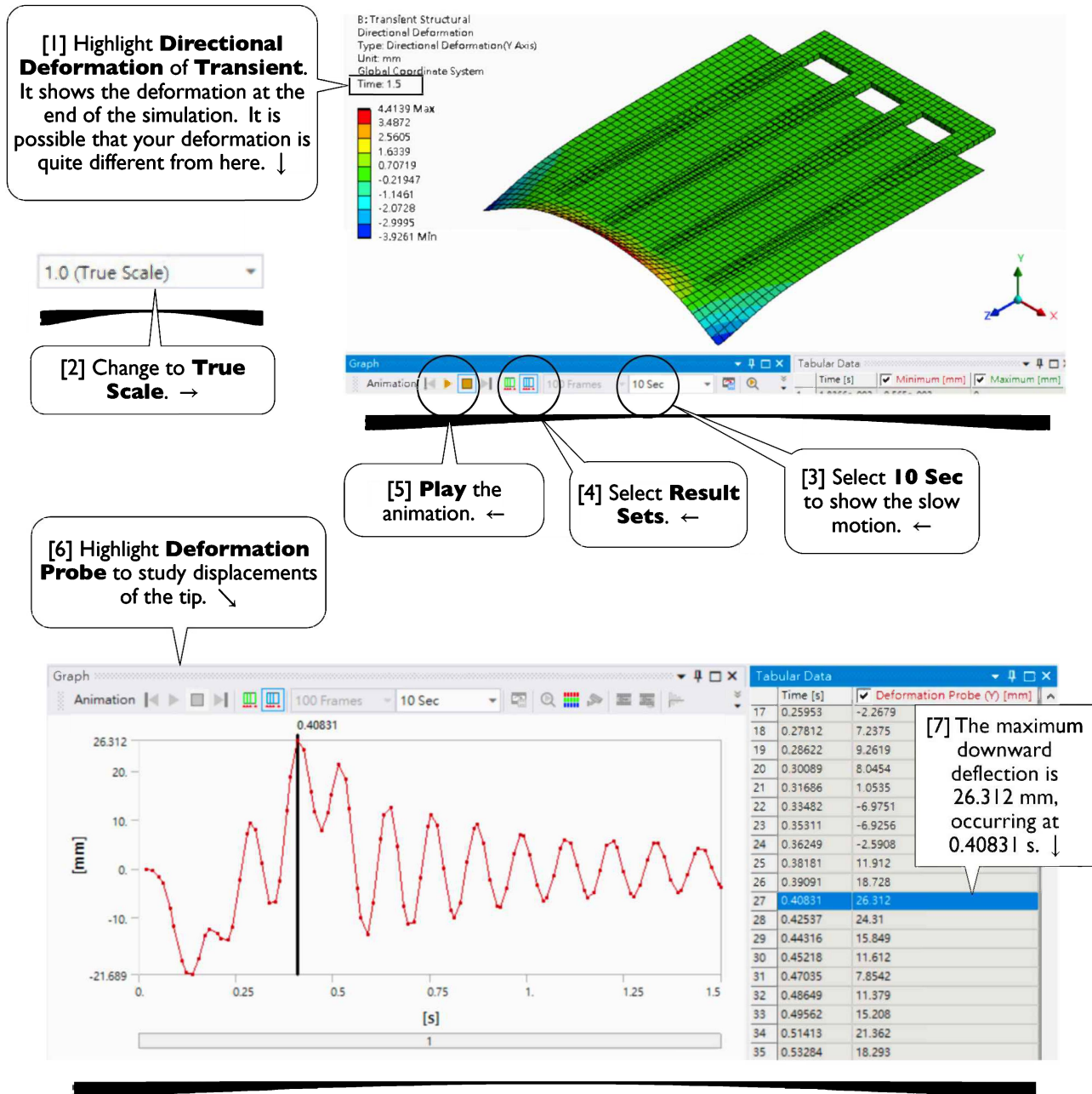
*** LOAD STEP      1  SUBSTEP  172  COMPLETED.    CUM ITER =
*** TIME = 1.48416    TIME INC = 0.915697E-02  NEW TRIANG
*** RESPONSE FREQ = 8.380    PERIOD= 0.1193    PTS/CYC =
*** AUTO TIME STEP: NEXT TIME INC = 0.91798E-02  INCREASED (FACTOR = 1.0025)

*** LOAD STEP      1  SUBSTEP  173  COMPLETED.    CUM ITER = 173
*** TIME = 1.49334    TIME INC = 0.917982E-02  NEW TRIANG MATRIX
*** RESPONSE FREQ = 8.429    PERIOD= 0.1176    PTS/CYC = 13.
*** AUTO TIME STEP: NEXT TIME INC = 0.66603E-02  DECREASED (FACTOR = 0.7255)

*** LOAD STEP      1  SUBSTEP  174  COMPLETED.    CUM ITER = 174
*** TIME = 1.50000    TIME INC = 0.666031E-02  NEW TRIANG MATRIX
*** RESPONSE FREQ = 8.707    PERIOD= 0.1148    PTS/CYC = 17.

```

12.2.6 View the Results



Observation

[8] The maximum deflection 26.312 mm [7], occurring at 0.40831 seconds, is about 50% more than that of the static deflection, which is 17.234 mm (12.2.4[6], page 434). The vibration damps out fast and reduces to about 5 mm in 1.5 sec. The response frequency is about 9 Hz (12.2.5[19], last page). Workbench automatically adjusts the time step according to the response frequency. Next, we want to investigate the maximum stress and its location. ↵



[9] With **Solution of Transient** highlighted, select **Probe/Stress**. →

[10] Select the glass body (Using body selection filter). ↓

Details of "Stress Probe"

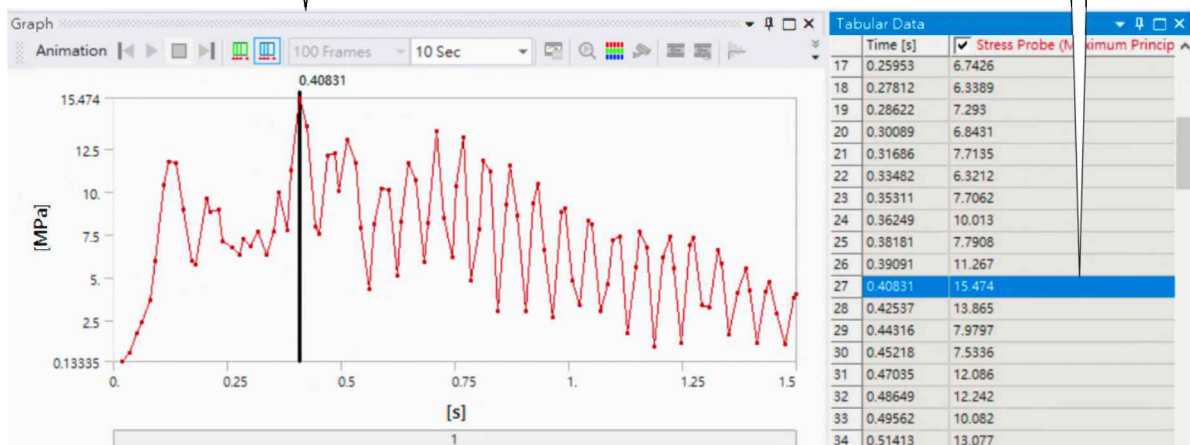
Definition	
Type	Stress
Location Method	Geometry Selection
Geometry	1 Body
Suppressed	No
Options	
Result Selection	Maximum Principal
Display Time	End Time
Spatial Resolution	Use Maximum
Results	
Maximum Value Over Time	
Maximum Principal	
Minimum Value Over Time	
Maximum Principal	
Information	

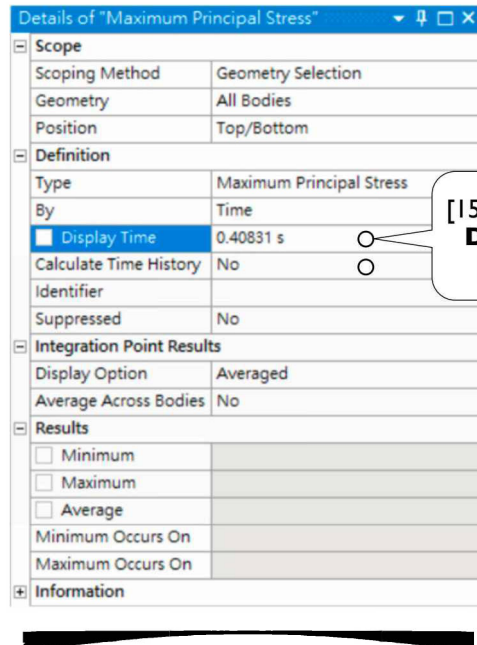
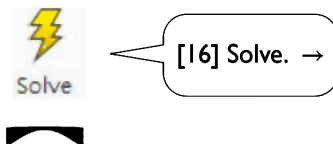
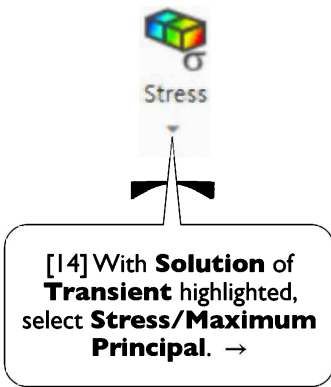


[11] Solve. ←

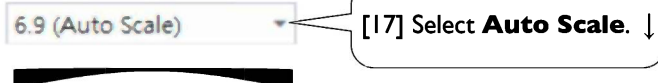
[12] Highlight **Stress Probe** to examine the history of maximum principal stress. →

[13] The maximum principal stress of the glass over the time is 15.474 MPa, occurring at 0.40831 seconds, when the maximum deflection occurs (also see [7], last page). ↙

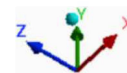
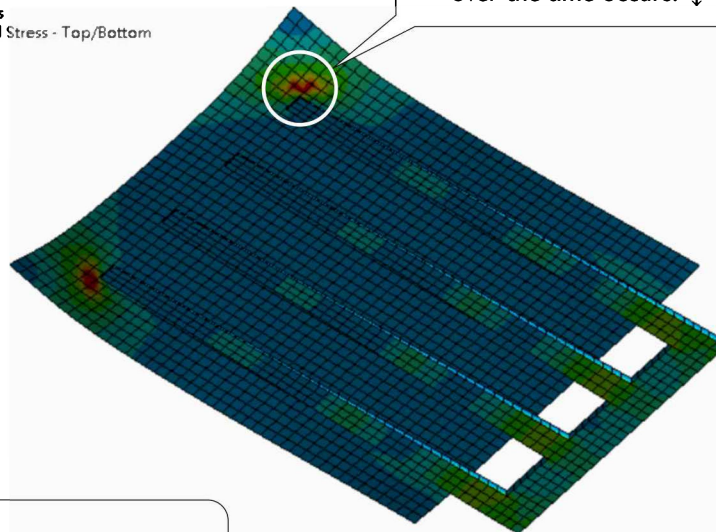
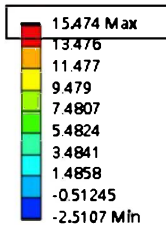




[15] Type 0.40831 (s) for **Display Time** (see [13], last page). ✓



B: Transient Structural
Maximum Principal Stress
Type: Maximum Principal Stress - Top/Bottom
Unit: MPa
Time: 0.40831

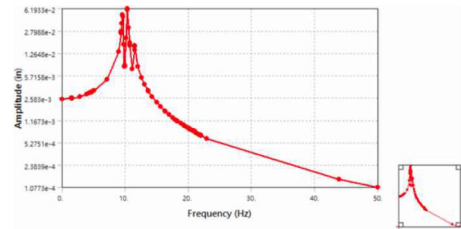


Wrap Up

[19] Save the project and exit Workbench. #

Section 12.3

Harmonic Response Analysis: Two-Story Building



12.3.1 About the Two-Story Building

[1] In this section, we will demonstrate the procedure of a harmonic response analysis (12.1.4[5], page 427). The two-story building (7.3, 11.2) is used again to demonstrate the procedures.

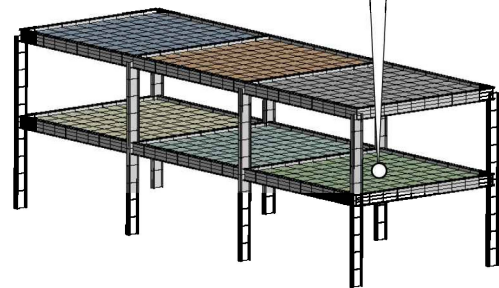
Harmonic Response Analysis

In 11.2.3[13] (page 397), we mentioned that the rhythmic loading on the floor may cause a safety issue. Is "dancing on the floor" really an issue? Since the building is designed to withstand a live load of 50 psf (lb/ft²), we will assume that a group of young people of 50 psf is dancing on a side-span floor deck [2] to simulate an asymmetric loading that will cause the building to side sway. The dancing is so hard that the young people generate a vertical periodical force of 10 psf, that is, the loading fluctuates from 40 psf to 60 psf.

Engineers usually don't consider "dancing" as a serious issue. Let's look at a more realistic engineering consideration. Imagine that an electric motor (or any rotatory machine) is installed on the floor deck [2]. The operational speed of the machine is 3000 rpm. When started up, the machine's speed increases from zero up to 3000 rpm. Is the vibration caused by the rotatory machine an issue?

In this section, we will perform a harmonic response analysis to answer these questions. →

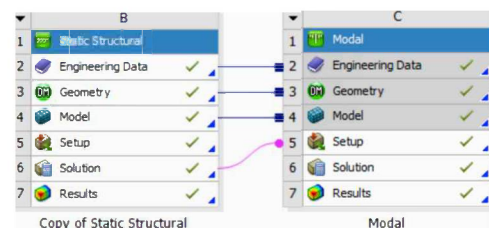
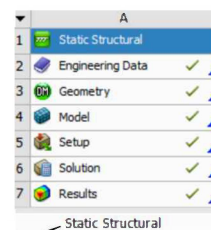
[2] Harmonic loads are applied on this floor deck. #



12.3.2 Perform Modal Analysis

[1] Launch Workbench. Open the project **Building**, which was saved in Section 11.2. →

[2] In this section, we want to reuse this system. Remember that the model in this system has no diagonal members. ↵



[3] From **Toolbox**, drag the **Modal** analysis system and drop here (**Solution**). →

Static Structural Modal

[4] Before a harmonic analysis, a modal analysis is usually needed. Another reason for performing a modal analysis is to investigate each vibration mode. Click any clickable cell in this system to start up **Mechanical**. ✓

Copy of Static Structural Modal

[5] Turn on **Large Deflection**. ↘

[6] Highlight **Analysis Settings** of **Modal** and type 20 for **Max Modes to Find**. ✓

Details of "Analysis Settings"	
Options	
Max Modes to Find	20
Limit Search to Range	No
Spin Softening	Program Controlled
Solver Controls	
Damped	No
Solver Type	Program Controlled
Rotordynamics Controls	
Output Controls	
Analysis Data Management	

[7] Select the **in-lbm-lbf-s** unit system and solve the model. ↙

Tabular Data		
	Mode	Frequency [Hz]
1	1.	1.5188
2	2.	4.0914
3	3.	4.1122
4	4.	4.8765
5	5.	9.3552
6	6.	9.5907
7	7.	9.9077
8	8.	10.333
9	9.	10.721
10	10.	11.499
11	11.	13.632
12	12.	16.308
13	13.	17.538
14	14.	18.227
15	15.	19.299
16	16.	20.091
17	17.	20.828
18	18.	21.181
19	19.	21.654
20	20.	21.934

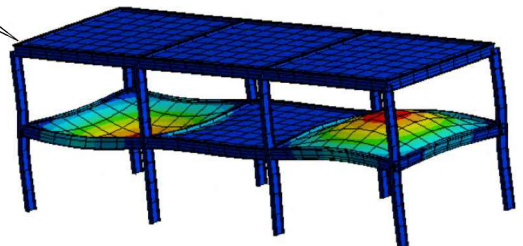
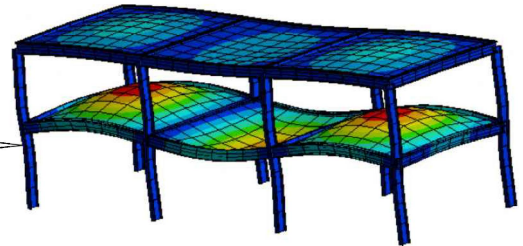
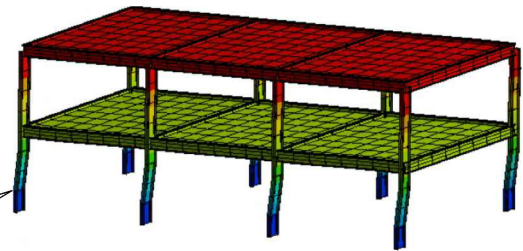
[8] Note that these frequencies are the same as those in 11.2.3[12], page 397. Create mode shape results. Explore each mode, particularly the 1st, 6th, and 8th modes [9-11]. These modes are important in our study. ↓

[9] This is the 1st mode (1.52 Hz). Remember to use **Auto Scale** and turn on **Display/Thick Shells and Beams**. ↓

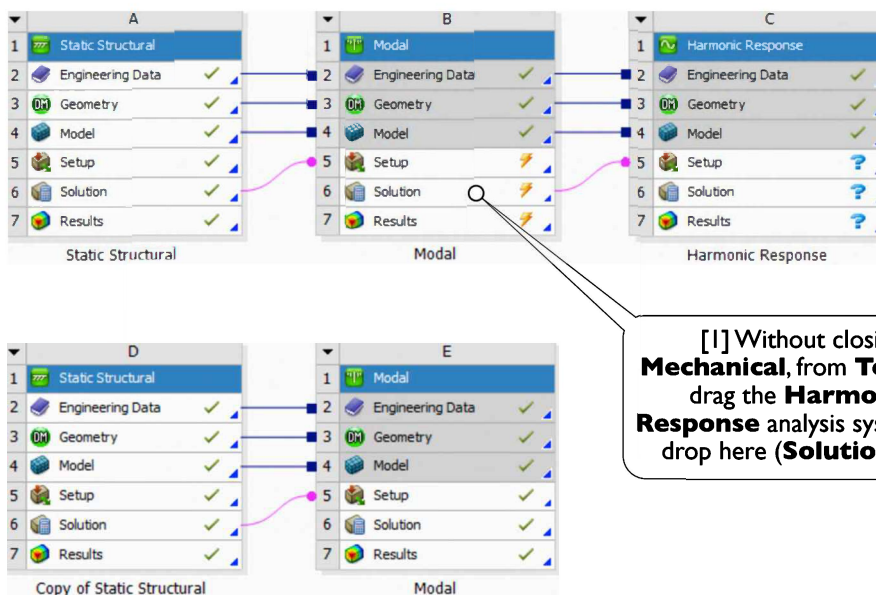
[10] The 6th mode (9.59 Hz). ↓

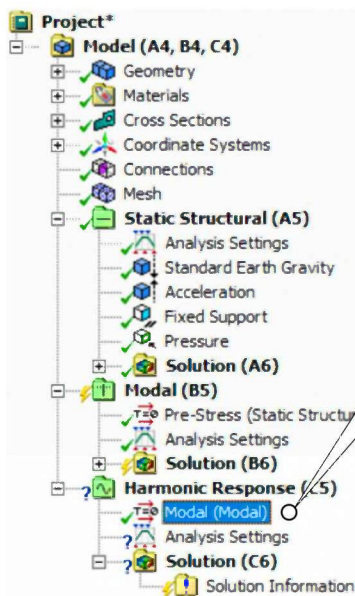
[11] The 8th mode (10.33 Hz). ↓

[12] Remember that we are studying the effect of dancing and the rotatory machine. The dancing frequency is close to the 1st mode (1.52 Hz); that's why we pay attention to this mode. For the rotatory machine, we are concerned about the vibrations of the floor deck (12.3.1[2], page 441) in the vertical direction; that's why we pay attention to the 6th and 8th modes. #



12.3.3 Perform Harmonic Response Analysis

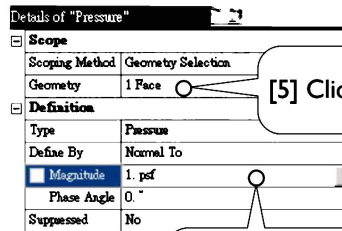




[2] Note that we are performing a prestressed harmonic response simulation; i.e., it includes all environment conditions defined in the **Static Structural** and **Modal** environments. Also note that the **Acceleration** (in **Static Structural**) should have been deleted; however, the results would have almost no differences (see 11.2.5[4], page 400).

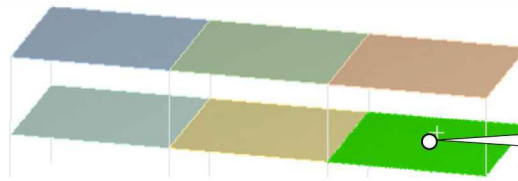


[3] With **Harmonic Response** highlighted, insert a **Pressure**. ↓

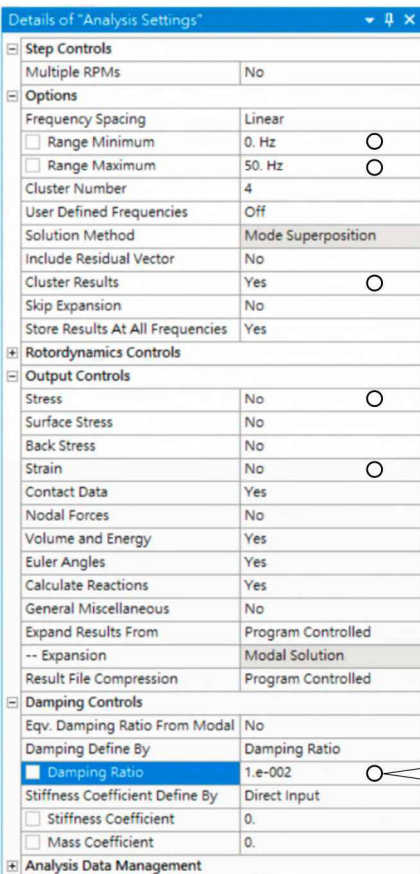


[5] Click **Apply**. ↓

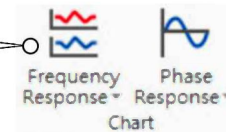
[6] Select the **ft-lbm-lbf-s** unit system and type **1 (psf)** for **Magnitude**. ✓



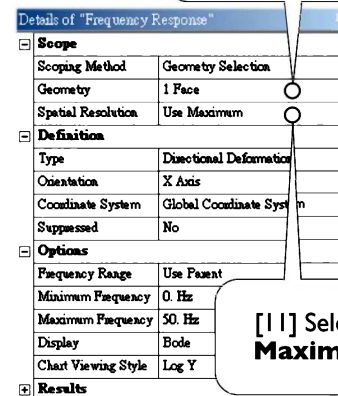
[4, 9] Select this face. ↑ ↓



[8] With **Solution of Harmonic Response** highlighted, select **Frequency Response/Deformation**. ↗

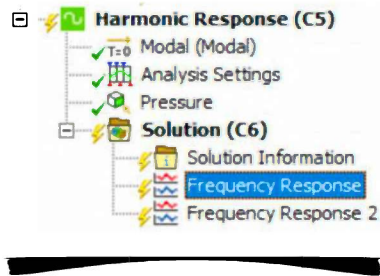


[10] Click **Apply**. ↓

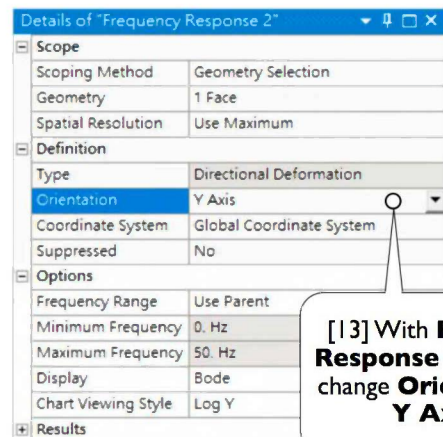


[11] Select **Use Maximum**. ←

[7] Set up **Analysis Settings of Harmonic Response** like this. We turn off (calculation of) **Stress** and (calculation of) **Strain** to save both runtime and disk space. It saves about 1 GB of disk space. ↗

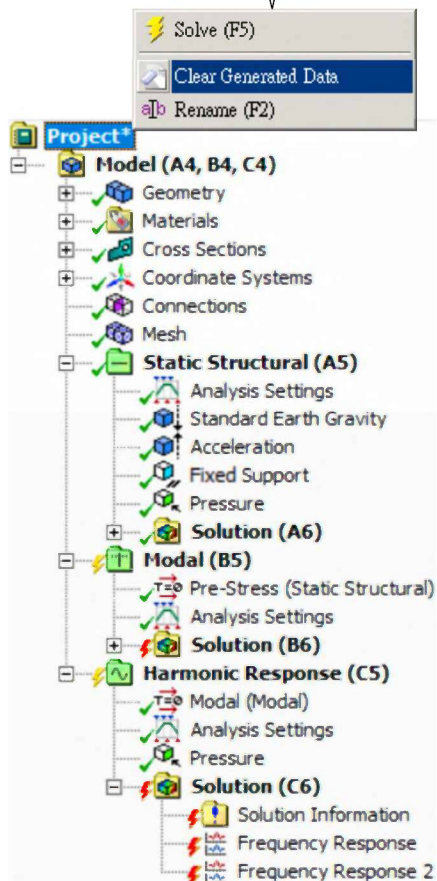


[12] Right-click **Frequency Response** and select **Duplicate** to create **Frequency Response 2**. →



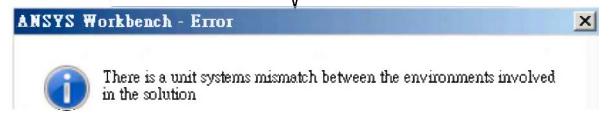
[13] With **Frequency Response 2** highlighted, change **Orientation** to **Y Axis**. ↓

[16] A way to tackle the "unit systems mismatch" error is to right-click **Project** and select **Clear Generated Data** and re-run the project. ↘



[14] Solve the model. If you run into a "unit systems mismatch" error, see [15-16]. ↓

[15] You may run into a "unit systems mismatch" error. ↘



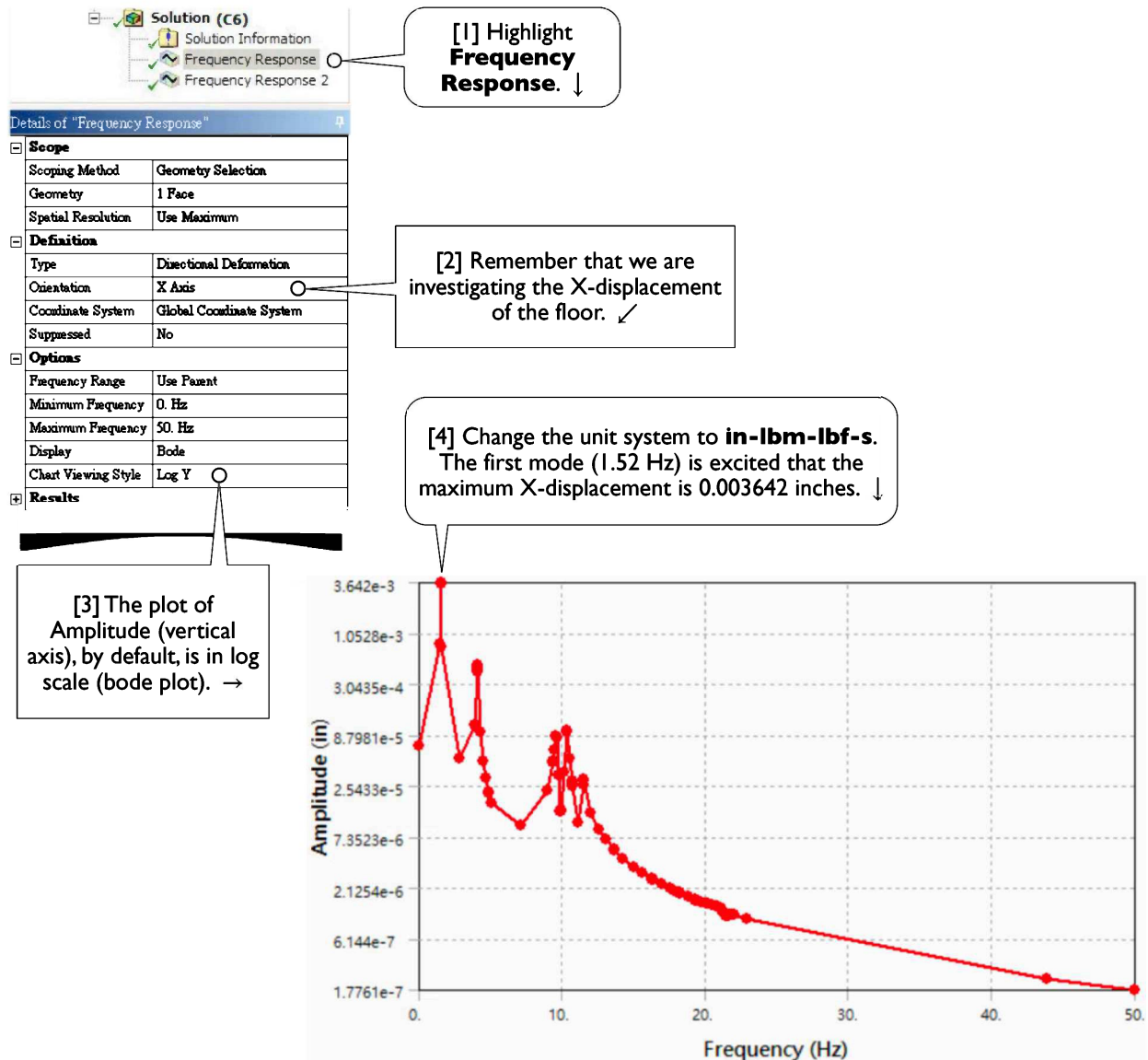
Why Unit Systems Mismatch?

[17] You use the length unit **inch** to solve the model in 12.3.2[7] (page 442; the resulting data are then stored in inches) and then use the length unit **ft** to perform the harmonic response analysis in [14] (see [6], last page). Using different unit systems might cause a unit systems mismatch.

Here, we've demonstrated a way to tackle the "unit systems mismatch" error [16]. Normally, Workbench should be able to maintain the consistency of the unit system no matter what unit system you are using; the "unit systems mismatch" shouldn't have happened in this case; it might be a bug.

A way to avoid the "unit systems mismatch" is that you always use the same unit system before solving the model. For example, in [14], select **in-lbm-lbf-s** unit system before clicking **Solve**. #

12.3.4 Effect of Dancing



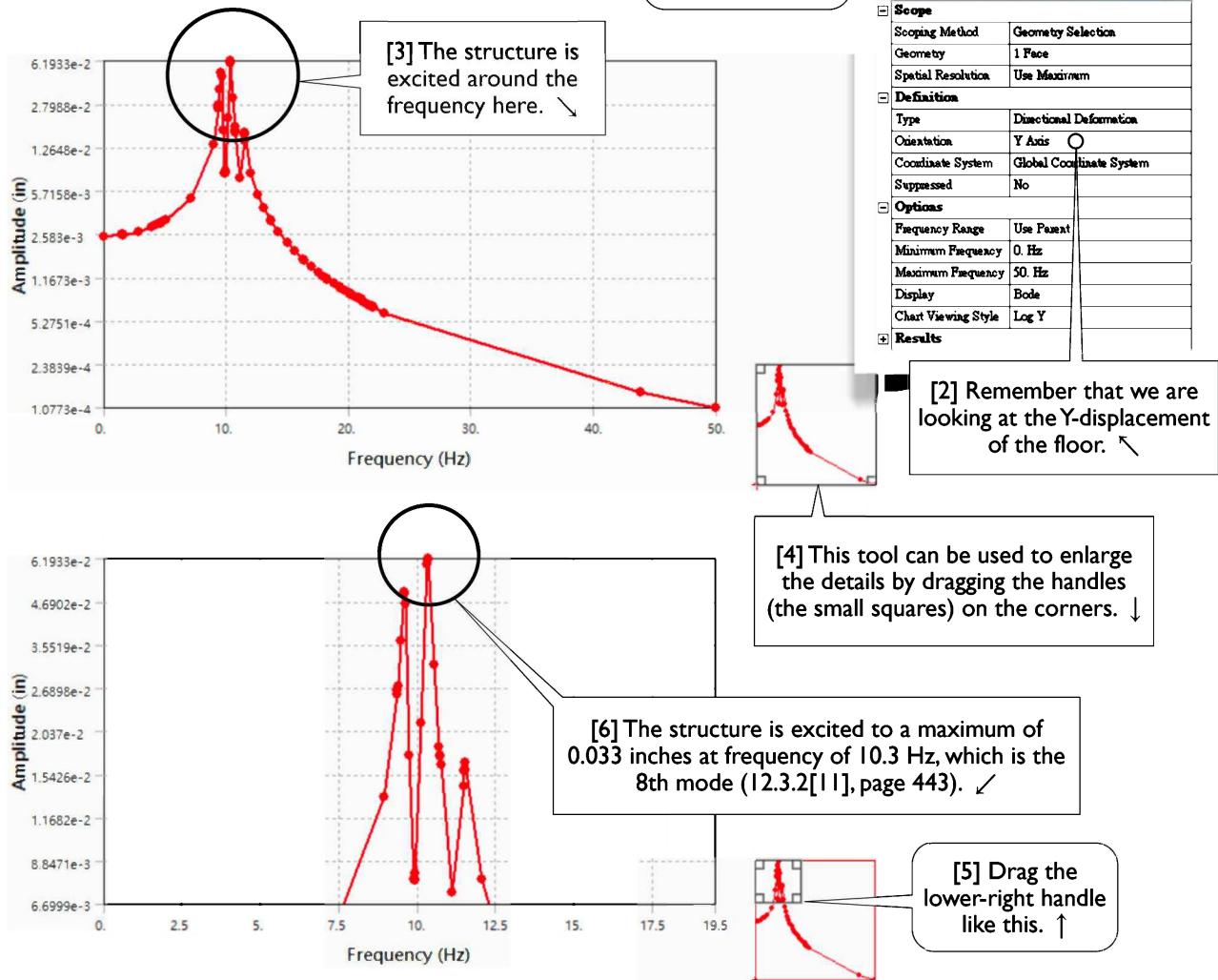
Interpretation of the Harmonic Response Plot

[5] The harmonic response plot [4] is an amplitude versus frequency plot. Because we are investigating the effect of the dancing, we should look at the frequencies that are less than 3 or 4 Hz (people don't dance faster than that). At dancing frequency of 1.52 Hz, the structure is excited such that the maximum X-displacement is 0.003642 in.

Remember that in 12.3.1[1] (page 441), the estimated dancing load is about 10 psf periodically. In 12.3.3[6] (page 444), we input 1 psf of harmonic load; therefore, the estimated response should be 10 times 0.003642 in, that is 0.03642 in. (0.925 mm). Obviously, this value is too small to be worried about.

We conclude that dancing is not an issue for this building. #

12.3.5 Effect of Rotatory Machine



Interpretation of the Harmonic Response Plot

[7] We were investigating the effect of the rotatory machine from 0 to 3000 rpm (50 Hz). We estimated that the amplitude of the harmonic load (of the electric motor) should be no more than 0.1 psf distributing on the floor (that totals to 40 lb). In 12.3.3[6] (page 444), we input 1 psf of harmonic load; therefore, the estimated response should be 0.1 times of the response shown in [3] (or [6]).

Although high frequencies do excite the floor, the values are very small. At frequency of 10.3 Hz, the excitation reaches a maximum of 0.0061933 in (0.1 times of 0.061933 in), or 0.157 mm. The value is too small to cause an issue.

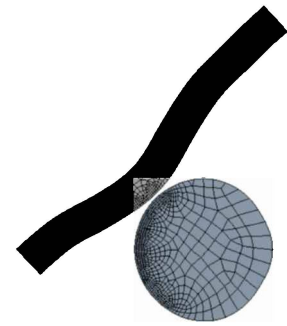
We conclude that the rotatory machine is safe for this building. ↓

Wrap Up

[8] Save the project and exit Workbench. #

Section 12.4

Disk and Block



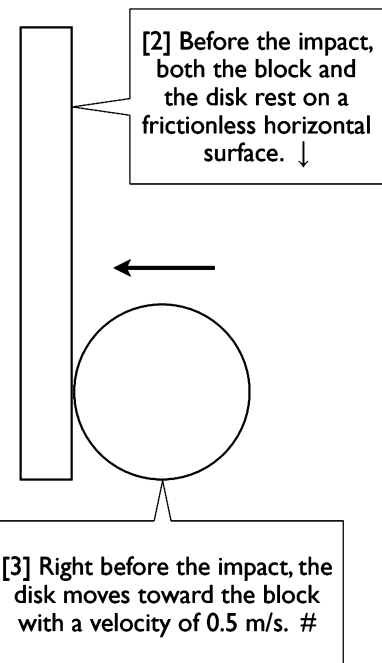
This exercise has two purposes: First, to demonstrate how to apply a simple initial condition, namely uniform velocity, on a body. Second, to show a limitation of **Transient Structural** analysis system for impact simulations and to motivate the students learning **Explicit Dynamics** in Chapter 15.

12.4.1 About the Disk and Block

[1] Consider a disk of radius of 40 mm and a block of 200 mm x 20 mm on a frictionless horizontal surface [2-3]; both have a thickness of 10 mm. Both are made of a very soft polymer of Young's modulus of 10 kPa, Poisson's ratio of 0.4, and mass density of 1000 kg/m³.

Right before the impact, the disk moves toward the block with a velocity of 0.5 m/s, and the positions of the disk and the block are as shown.

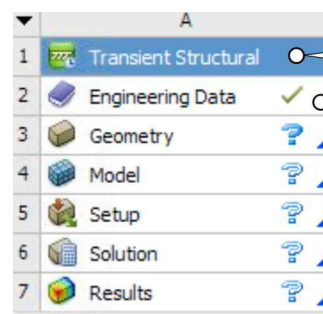
We purposely consider a very soft material (Young's modulus of 10 kPa) and a very slow-speed impact (velocity of 0.5 m/s) to relieve numerical difficulty. Increasing either of them would make the impact duration shorter, and in turn require a shorter integration time step (to find a solution). This is left as an exercise for you at the end of this chapter. →



12.4.2 Start Up

[1] Launch Workbench. Create a **Transient Structural** analysis system [2] by double-clicking it in **Toolbox**. Save the project as **Disk**.

Double-click **Engineering Data** to prepare material data [3]. →



[2] Create a **Transient Structural** analysis system. ↓

[3] Double-click **Engineering Data** to prepare material data. ↵

[4] Create a new material **Polymer**, and include **Density** and **Isotropic Elasticity**. ↘

[5] Type material properties (**Density, Young's Modulus, Poisson's Ratio**) as shown. Note that the SI units are used. ←

[6] Return to **Project Schematic**. #

Property	Value	Unit
Material Field Variables	Table	
Density	1000	kg m ⁻³
Isotropic Elasticity		
Derive from	Young's Modulus and Poisson's Ratio	
Young's Modulus	10000	Pa
Poisson's Ratio	0.4	
Bulk Modulus	16667	Pa
Shear Modulus	3571.4	Pa

12.4.3 Create Geometry in DesignModeler

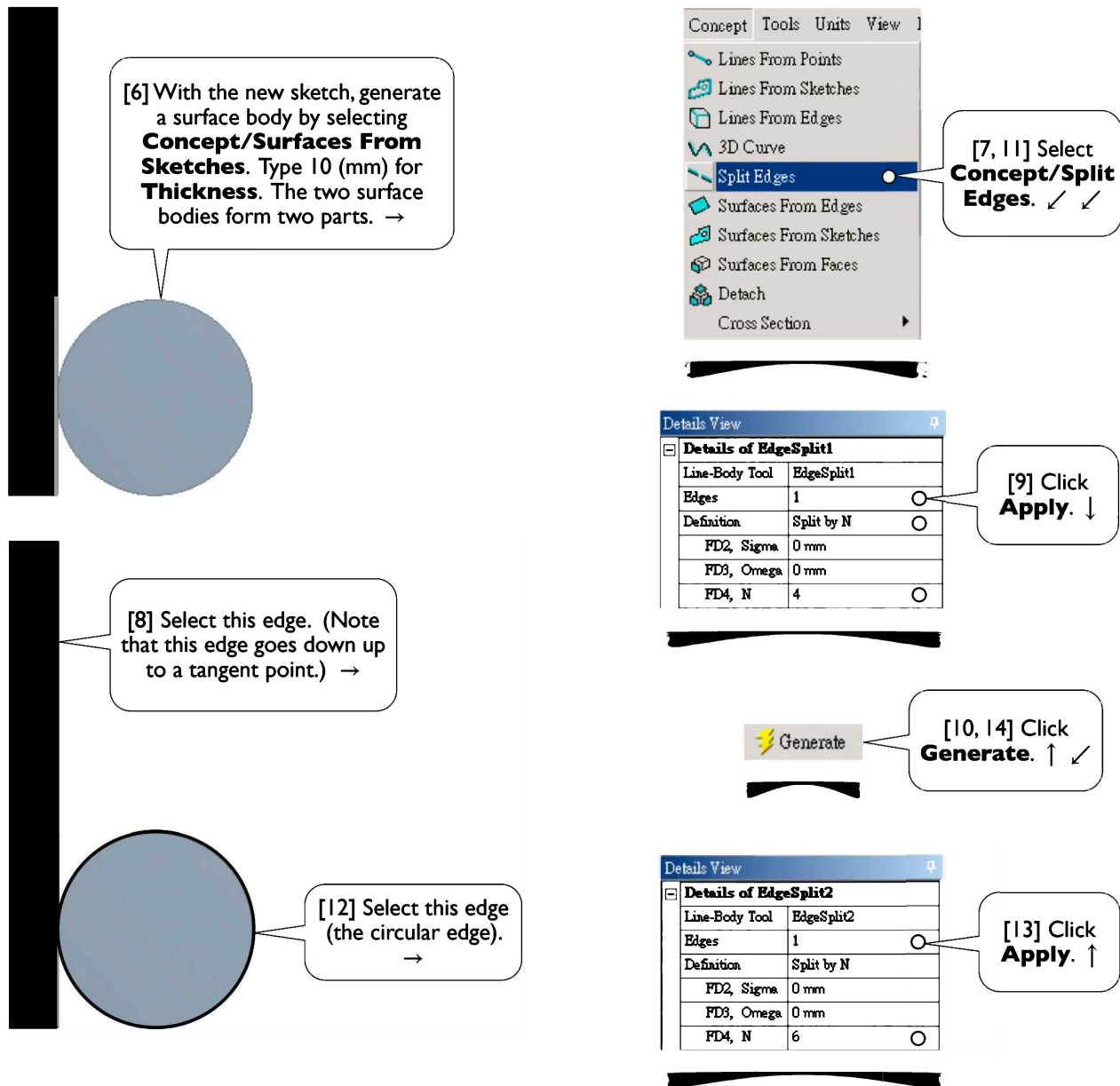
[1] Start up DesignModeler, Select **Millimeter** as the length unit. On **XYPlane**, draw a rectangle like this. →

[2] With the sketch, generate a surface body by selecting **Concept/Surfaces From Sketches**. Type 10 (mm) for **Thickness**. →

[3] On **XYPlane**, create a new sketch (**Sketch2**) and draw a circle like this. Impose two **Tangent** constraints [4-5]. ↓

[4] Tangent. ←

[5] Tangent. ↙

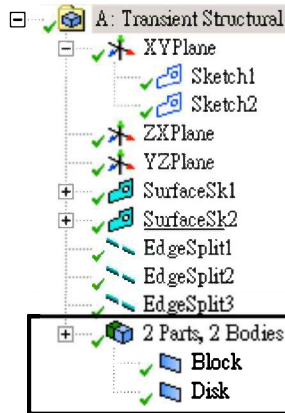


Why Split Edges?

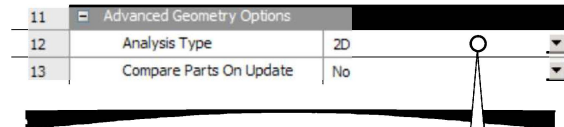
[15] The purpose of splitting the edges is that we may specify a mesh density for each segment. We need finer mesh near the contact region (see 12.4.4[7-9], page 452).

How to Make Sure an Edge is Successfully Split?

Move your mouse on a split edge, which is then highlighted. Go through each split edge to make sure an edge is successfully split. ↵

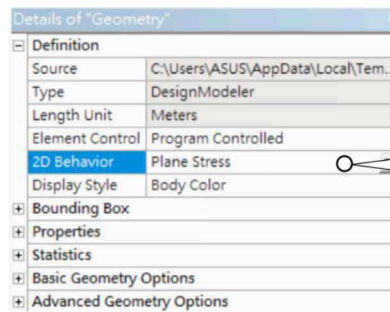
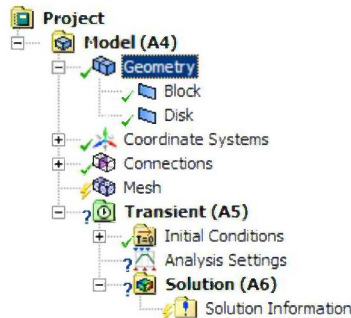


[16] Rename the two surface bodies like this. →

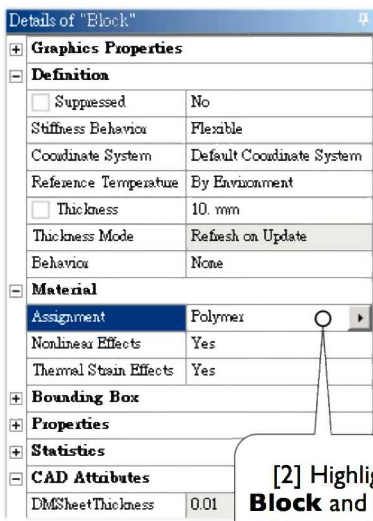


[17] Close DesignModeler. Set **Analysis Type** to **2D** (3.1.4[4-6], page 112). #

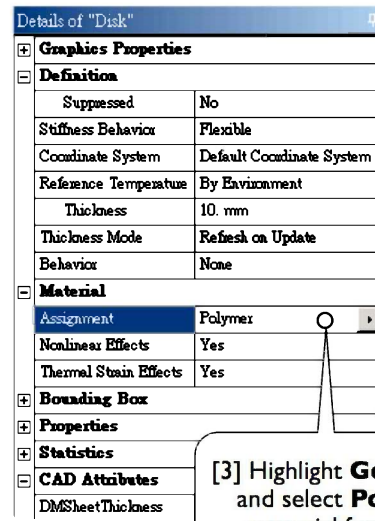
12.4.4 Simulation in Mechanical



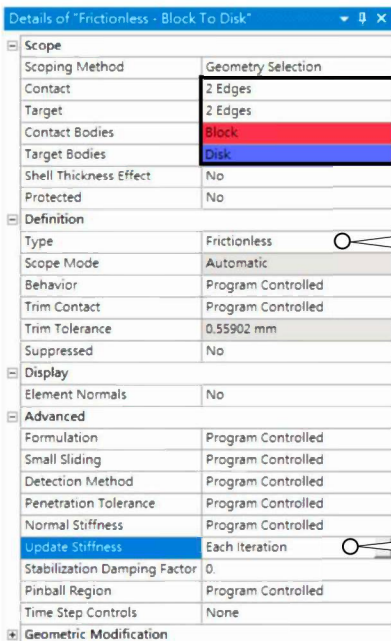
[1] Start up **Mechanical** and select the **mm-kg-N-s** unit system. Highlight **Geometry**, and make sure **Plane Stress** is selected for **2D Behavior**. ✓



[2] Highlight **Geometry/Block** and select **Polymer** as the material for the body. →



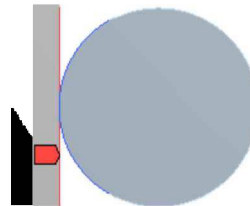
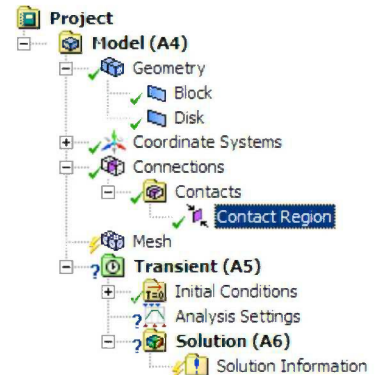
[3] Highlight **Geometry/Disk** and select **Polymer** as the material for the body. ↵



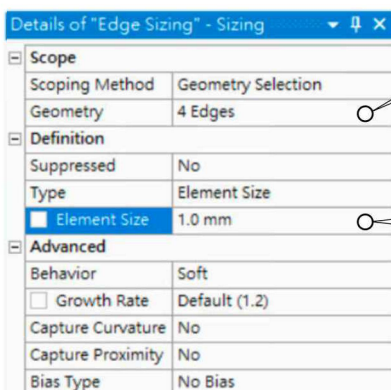
[4] Highlight **Connection/Contacts/Contact Region**. The contact region is correctly detected by Workbench. ↓

[5] Change the contact type to **Frictionless**. ↓

[6] Update stiffness at each equilibrium iteration (see 13.1.1.1 [1], page 479). It usually facilitates convergence when contacts are involved but needs more computing time. ↓

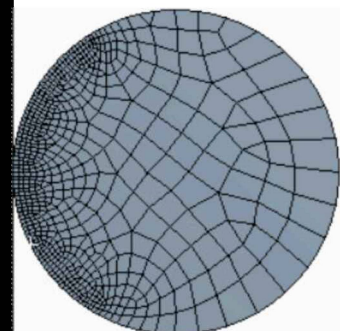


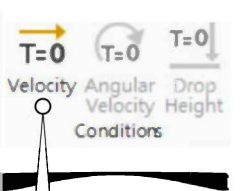
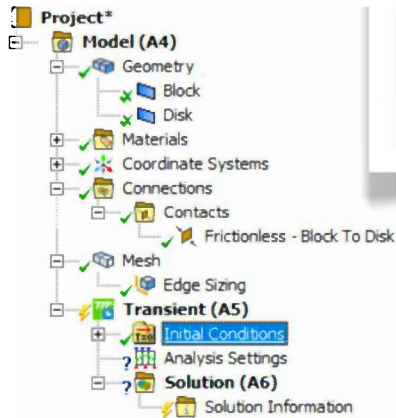
[7] With **Mesh** highlighted, select **Mesh/Sizing**. ↓



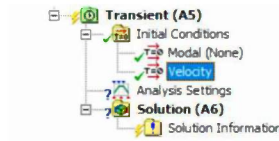
[8] Control-select the four edges (using the edge filter) in the contact region (two in the block and two in the disk, see [4]). ↓

[9] Type **1** (mm) for **Element Size**. Generate mesh. ↵

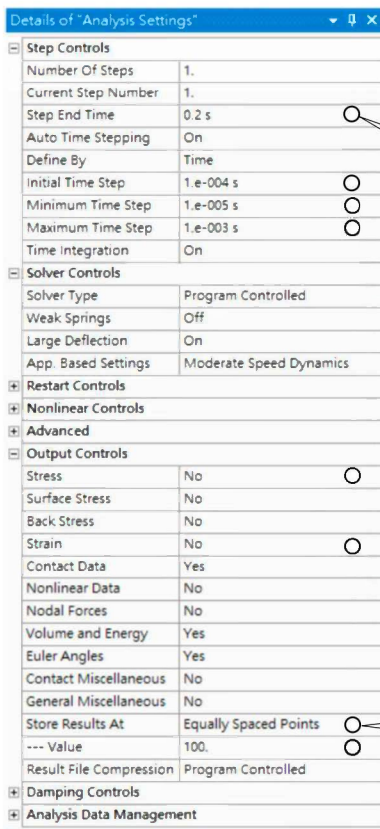
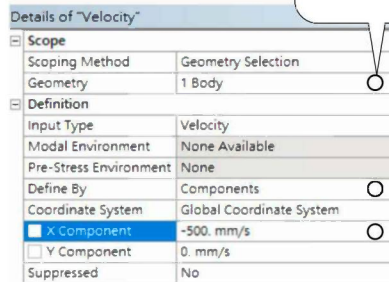




[10] With Transient/Initial Conditions highlighted, click Velocity. →

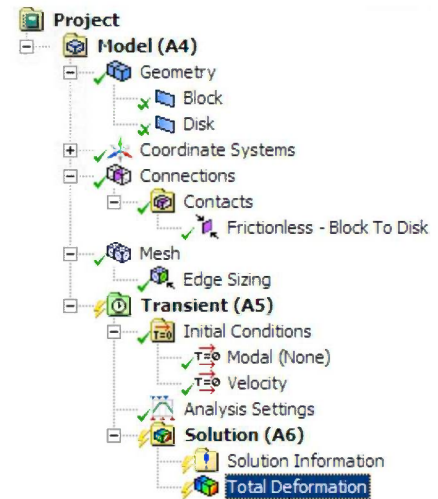


[11] Select the disk body (using body filter) and specify the velocity. ✓

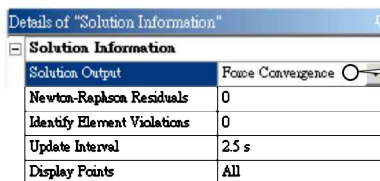


[12] Highlight Analysis Settings. Set up Step Controls like this. ↓

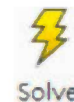
[13] Set up Output Controls like this to reduce data storage space and computing time. →



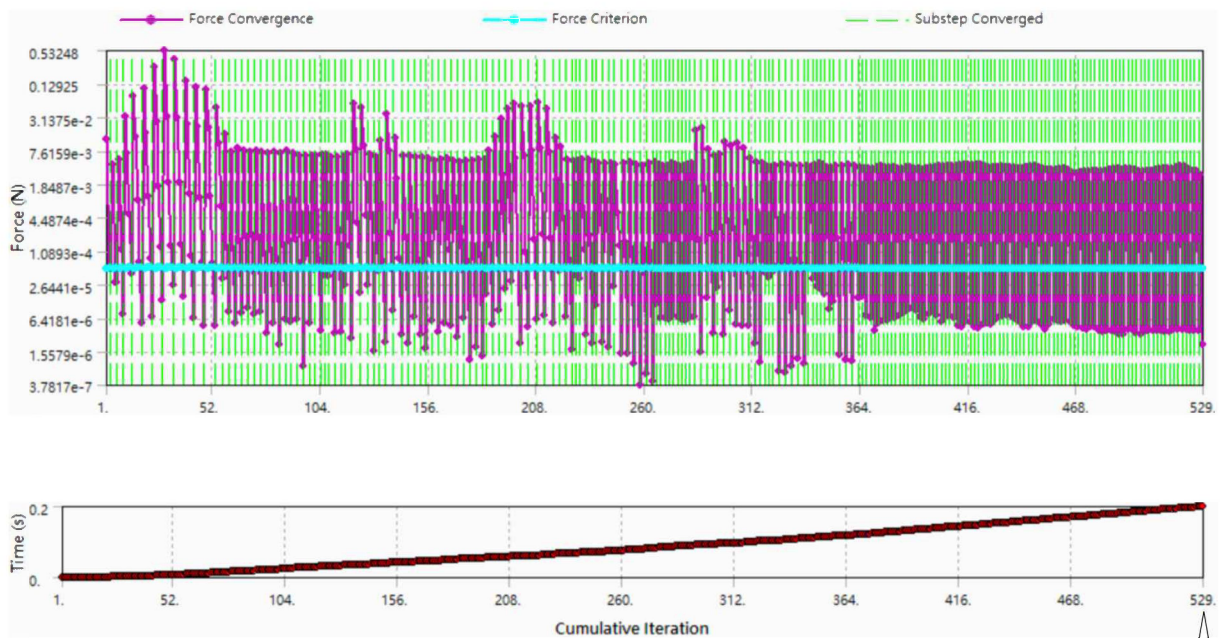
[14] Insert a Total Deformation. ✓



[15] With Solution/Solution Information highlighted, select Force Convergence to watch how the solution proceeds. →



[16] Click Solve. ↩



[17] It takes many time steps to complete the simulation. #

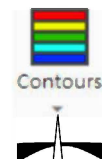
12.4.5 Animate the Impact



[1] Highlight **Total Deformation**. →



[2] Select **1.0 (True Scale)**. ↓

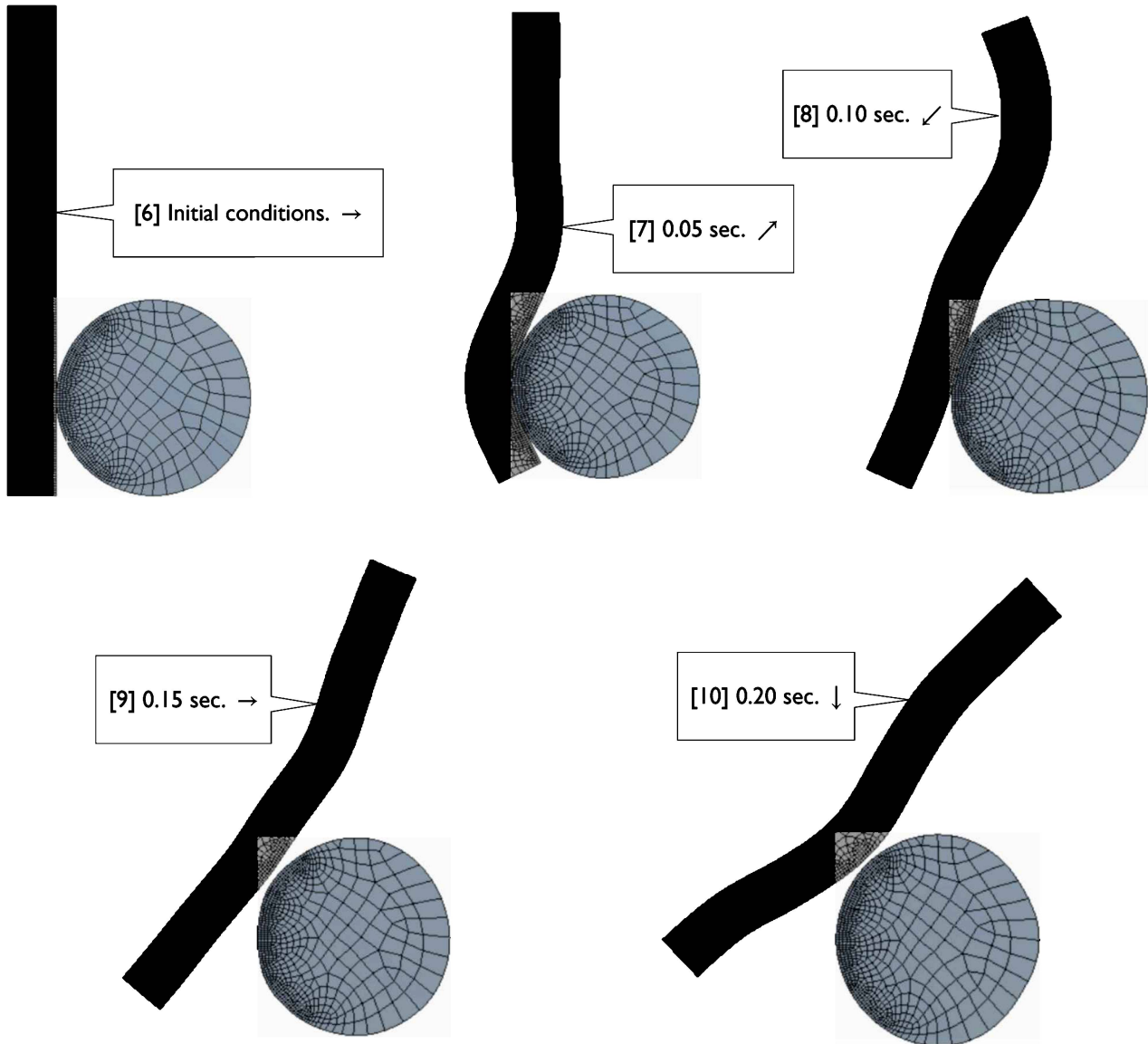


[3] Select **Result/Contours/Solid Fill**. Color contours are not important here. ↓



[5] Click **Play**. ↵

[4] Select **Result Sets**. ←



How to Obtain a Snapshot?

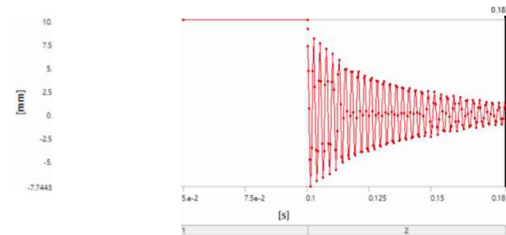
[11] Highlight **Total Deformation**, type the time for **Display Time**, and click **Solve**. ↓

Wrap Up

[12] Save the project and exit Workbench. #

Section 12.5

Guitar String



This exercise has two purposes: First, to demonstrate how to set up an initial condition using the result of a static simulation. Second, to show the transient behavior of a guitar string, as a sequel of Section 11.4.

12.5.1 About the Guitar String

[1] As mentioned, in 11.4.3[1] (page 413), if you pluck a string, you will produce a tone made up of all harmonic modes. It is the harmonic mixes that differentiate the sound of one music instrument from others. A good guitar player knows very well that he/she can control the quality of sound by plucking the string at different locations. We will explore this phenomenon in this section, using the guitar string introduced in Section 11.4.

The string is plucked at the middle, quarter, and eighth points respectively, and its transient responses are observed. More precisely, the string is applied a vertical displacement of 10 mm at respective locations, and then instantaneously released to produce vibrations. The vibrations will eventually reach a steady state, that is, free vibrations, after a certain duration of time. It, however, is the transient vibrations that impress our ears most. #

12.5.2 Using the Result of Static Analysis as Initial Condition

[1] Applying the vertical displacement requires a **Static Structural** analysis. Releasing the string to produce vibrations requires a **Transient Structural** analysis using the results of the static analysis as an initial condition. In a **Transient Structural**, it is possible to specify the first step as a static simulation. The results become the initial condition of the next step of transient simulation. This two-step method has become a standard procedure, when the simulation requires a static simulation as an initial condition.

Before we demonstrate the two-step method, let's clear up some important concepts. In transient dynamic simulation, each time step needs an initial condition to carry it on. The results of the last time step become the initial condition of the next time step. The initial condition, more specifically, is the position and velocity of each node.

In a **Transient Structural** simulation, Workbench allows you to turn off/on **Time Integration** ([2], next page). Turning **Time Integration** off means turning the dynamic effects (1.1.10[1], page 21) off, and Eq. 12.1.4(1) (page 426) reduces to Eq. 1.3.1(1) (page 35), i.e., a static simulation. You can turn on/off **Time Integration** in any load step.

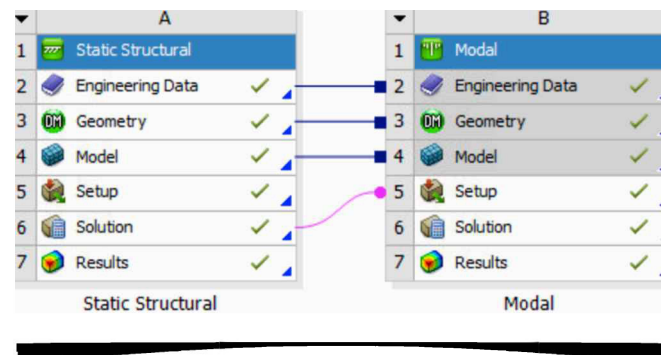
Make sure you don't confuse load steps (or simply called steps), substeps, and equilibrium iterations. Multiple load steps can be created when you need to specify different **Analysis Settings** for each step. Each step is further divided into substeps (also called time steps) for two reasons: First, in a transient dynamic simulation, each substep is a time integration step. Second, in a nonlinear simulation, a substep must be small enough to achieve convergence within that substep. For nonlinear simulations (either dynamic or static), each substep may need multiple equilibrium iterations to achieve convergence. ↵

Details of "Analysis Settings"	
Step Controls	
Number Of Steps	2.
Current Step Number	1.
Step End Time	0.1 s
Auto Time Stepping	Off
Define By	Time
Time Step	5.e-002 s
Time Integration	Off

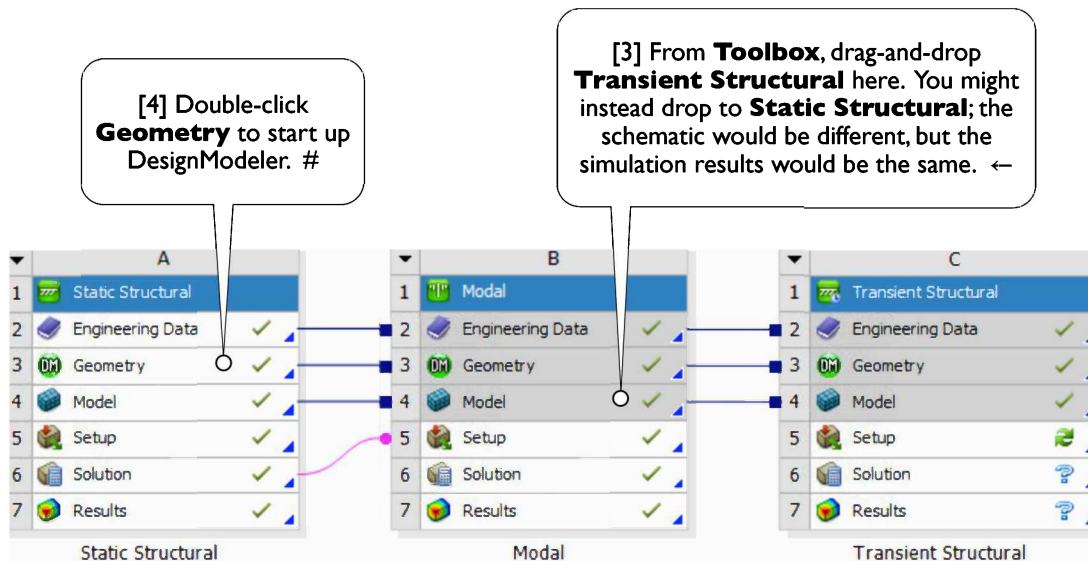
[2] In a **Transient Structural** simulation, if **Time Integration** is turned off (i.e., dynamic effects are turned off), it reduces to a static simulation. #

12.5.3 Resume the Project **String**

[1] Launch Workbench. Open the project **String**, which was saved in Section 11.4. →



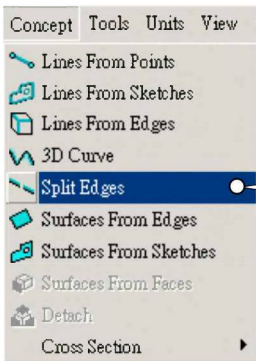
[2] This is the project schematic at the end of Section 11.4. ↓



[4] Double-click **Geometry** to start up DesignModeler. #

[3] From **Toolbox**, drag-and-drop **Transient Structural** here. You might instead drop to **Static Structural**; the schematic would be different, but the simulation results would be the same. ←

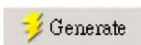
12.5.4 Modify Geometry in DesignModeler



[1] Select
**Concept/Split
Edges.** →

Details View		
Details of EdgeSplit1		
Line-Body Tool	EdgeSplit1	
Edges	1	<input type="radio"/>
Definition	Split by N	<input type="radio"/>
FD2, Sigma	0 mm	
FD3, Omega	0 mm	
FD4, N	8	<input type="radio"/>

[2] Select the
string. ✓

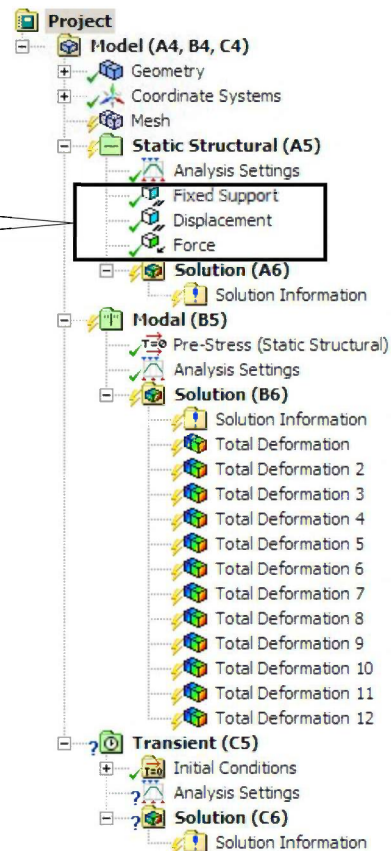


[3] Click
Generate. →

[4] Close DesignModeler. In **Project Schematic**, double-click any cells other than **Geometry** or **Engineering Data** to start up **Mechanical**. #

12.5.5 Set Up Environment Conditions

[1] Since we've modified the geometry, it is possible that the environment conditions are modified. If so, we need to correct them. In this case, however, the environment conditions do not need to be corrected. You should verify this yourself. ↵



A: Static Structural
Static Structural
Time: 1. s

A Force: 387.16 N
B Displacement
C Fixed Support

[2] Highlight **Static Structural**; the environment conditions should look like this. ✓

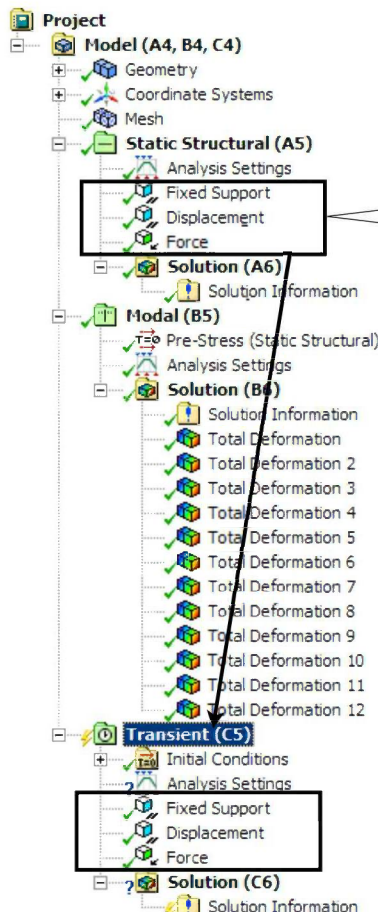


[3] Since the geometry is modified, the results need to be re-generated. Highlight **Modal** and click **Solve**. Workbench solves both **Static Structural** and **Modal**. →



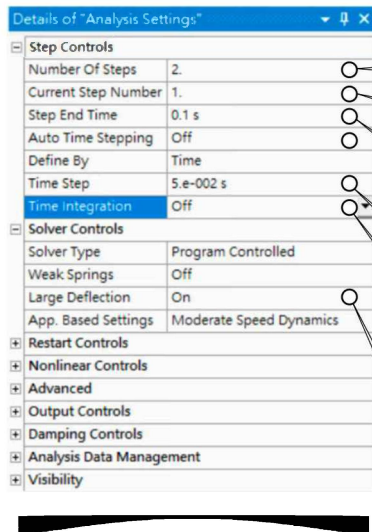
Tabular Data		
	Mode	Frequency [Hz]
1	1.	440.
2	2.	440.
3	3.	800.8
4	4.	880.
5	5.	880.
6	6.	1241.3
7	7.	1320.
8	8.	1320.
9	9.	1760.1
10	10.	1760.1
11	11.	2200.3
12	12.	2200.3

[4] The natural frequencies should be the same as before (11.4.2[12], page 412), with negligible numerical differences. ↓



[5] Control-select these three environment conditions in **Static Structural** and drag-and-drop them to **Transient**. #

12.5.6 Set Up Analysis Settings



[1] Highlight **Analysis Settings** of **Transient** and type 2 for total number of steps. ↓

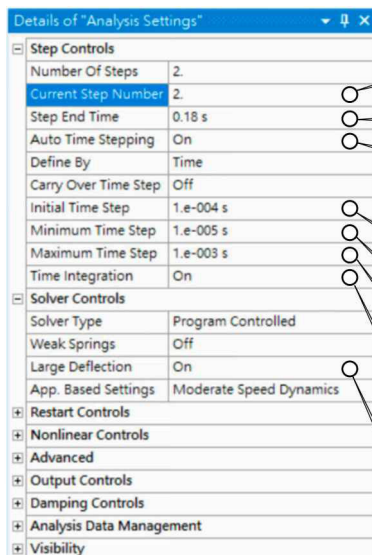
[2] We're now setting up the first load step. ↓

[3] The ending time of the first step is arbitrary. Here we type 0.1 (s). ↓

[4] The first load step is further divided into two substeps, each 0.05 s. We will explain this later. ↓

[5] Turn off **Time Integration** to perform a static simulation for this step. ↓

[6] Make sure **Large Deflection** is on. The prestress (387.16 N; 11.4.2[11], page 412) can be included only in a nonlinear simulation. ↓



[7] Type 2 (or click Step 2 in **Tabular Data**). We're now setting up the second load step. ↓

[8] The total time for transient simulation is 0.08 seconds ($0.18 - 0.10 = 0.08$). ↓

[9] Make sure **Auto Time Stepping** is on to let Workbench decide integration time steps according to response frequencies. ↓

[10] Set the initial time step to 0.0001 seconds. ↓

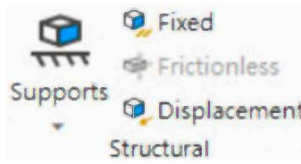
[11] Set the minimum time step to 1/10 of the initial time step. ↓

[12] Set the maximum time step to 10 times the initial time step. ↓

[13] Make sure **Time Integration** is on for the second step. ↓

[14] Make sure **Large Deflection** is on so that the prestress is included. #

12.5.7 Set Up Initial Condition



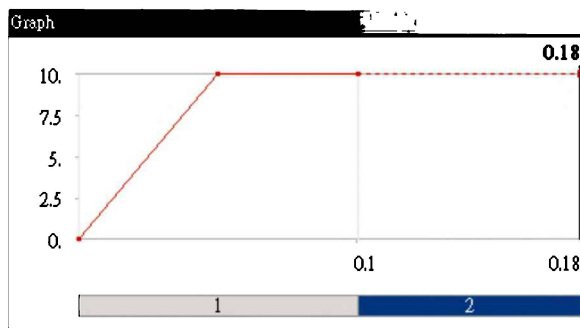
[1] With **Transient** highlighted, insert a **Displacement**.

Details of "Displacement 2"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Displacement
Base Excitation	No
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
Y Component	Tabular Data
Z Component	Free
Suppressed	No

[2] Select the middle point of the string, using the vertex filter if necessary. ↓

[3] Select **Tabular (Time)** for **Y Component**. ↓

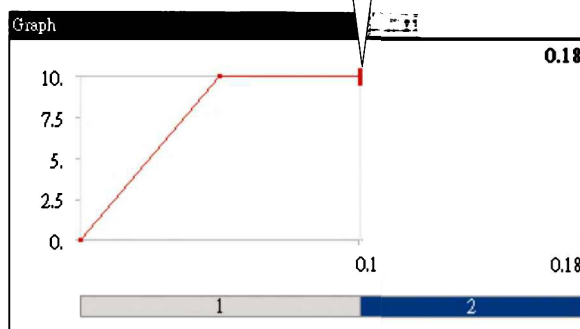


Tabular Data

Steps	Time [s]	Y [mm]
1	0.	0.
2	5.e-002	10.
3	0.1	10.
4	0.18	= 10.
*		

[4] Type tabular data like this (also see 12.2.5[9], page 436; the equal sign = is not relevant in this case). ↓

[6] The second step is deactivated (i.e., the displacement is released). #



Tabular Data

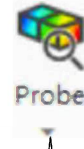
Steps	Time [s]	Y [mm]
1	0.	0.
2	5.e-002	10.
3	0.1	10.
4	0.18	= 10.
*		

[5] Select the second step and Right-click-select **Activate/Deactivate at this step!** ↑

12.5.8 Insert Result Objects and Solve the Model



[1] With **Solution of Transient** highlighted, insert a **Deformation/Total**. →



[2] Insert a **Probe/Deformation**. ✓

Details of "Deformation Probe"

Definition	
Type	Deformation
Location Method	Geometry Selection
Geometry	1 Vertex
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	Y Axis
Display Time	End Time
Spatial Resolution	Use Maximum
Results	
Maximum Value Over Time	
Y Axis	
Minimum Value Over Time	
Y Axis	
Information	

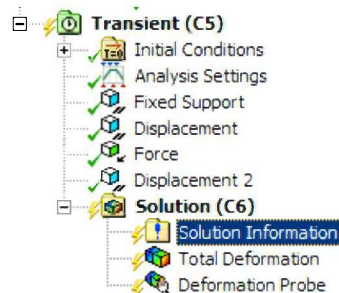
[3] Select the middle point of the string. ↓

[4] We want to see the Y-displacement. ↓

[5] Highlight **Solution/Solution Information** of **Transient**. Select **Force Convergence** to watch how Workbench solves the model. ↓

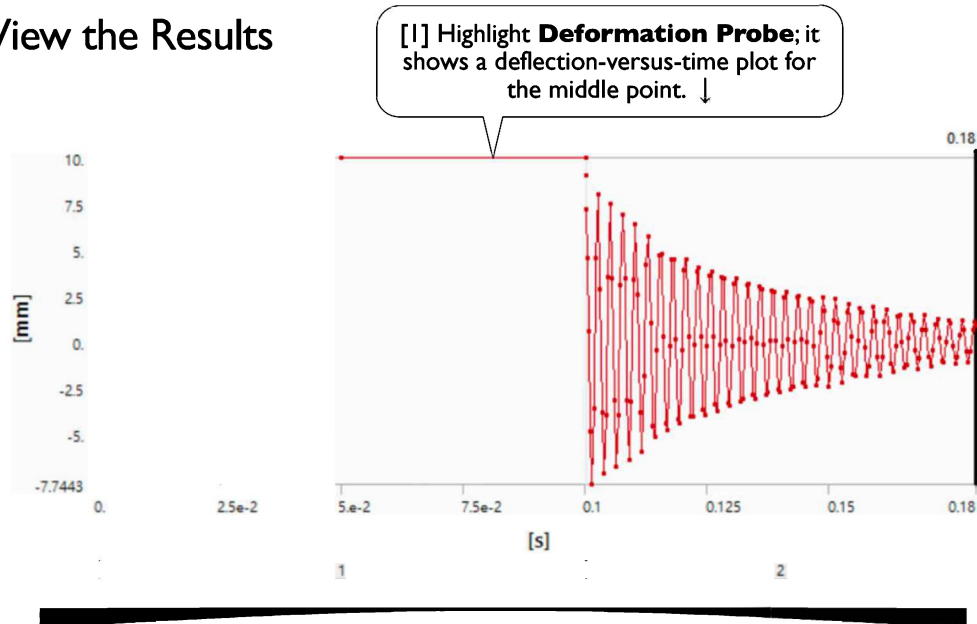
Details of "Solution Information"

Solution Information	
Solution Output	Force Convergence
Newton-Raphson Residuals	0
Identify Element Violations	0
Update Interval	2.5 s
Display Points	All
FE Connection Visibility	
Activate Visibility	Yes
Display	All FE Connectors
Draw Connections Attached To	All Nodes
Line Color	Connection Type
Visible on Results	No
Line Thickness	Single
Display Type	Lines

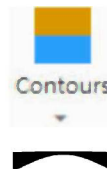


[6] Click **Solve**. #

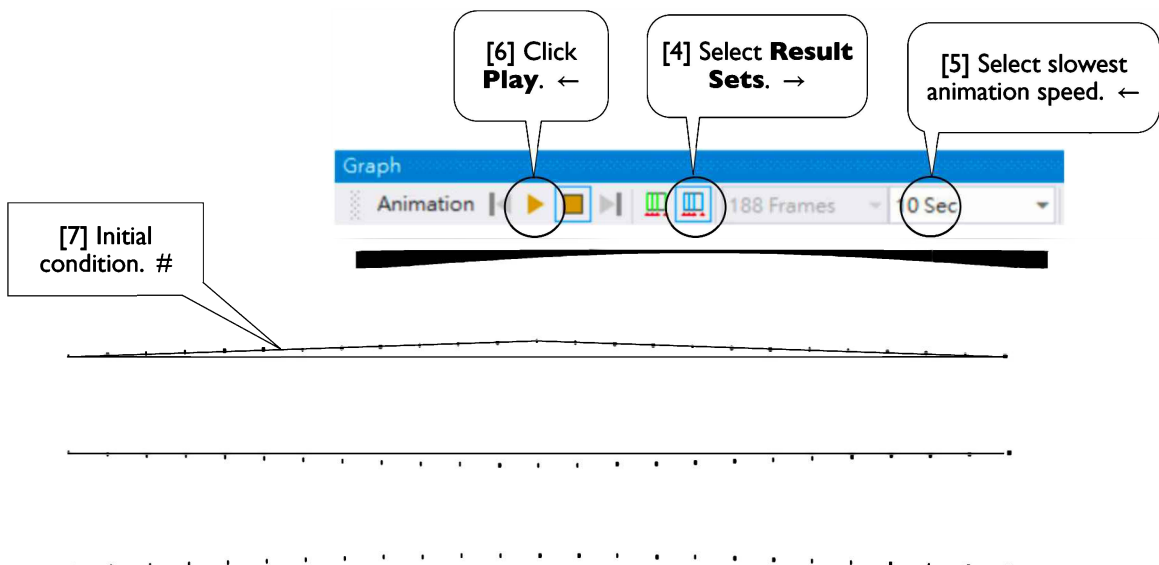
12.5.9 View the Results



[2] We are now preparing an animation. Highlight **Solution/ Total Deformation of Transient**. Make sure **Display/Style/Thick Shells and Beams** is off. →



[3] Select **Result/ Display/Contours/ Solid Fill**. Color contours are not important here. ✓



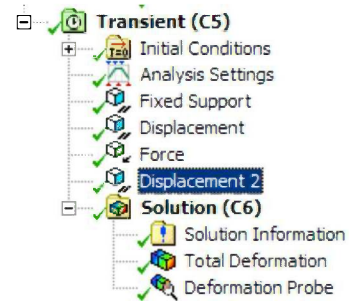
12.5.10 Pluck at Quarter Point

Details of "Displacement 2"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Displacement
Base Excitation	No
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
Y Component	Tabular Data
Z Component	Free
Suppressed	No

Displacement 2
Components: Free, 10, Free mm

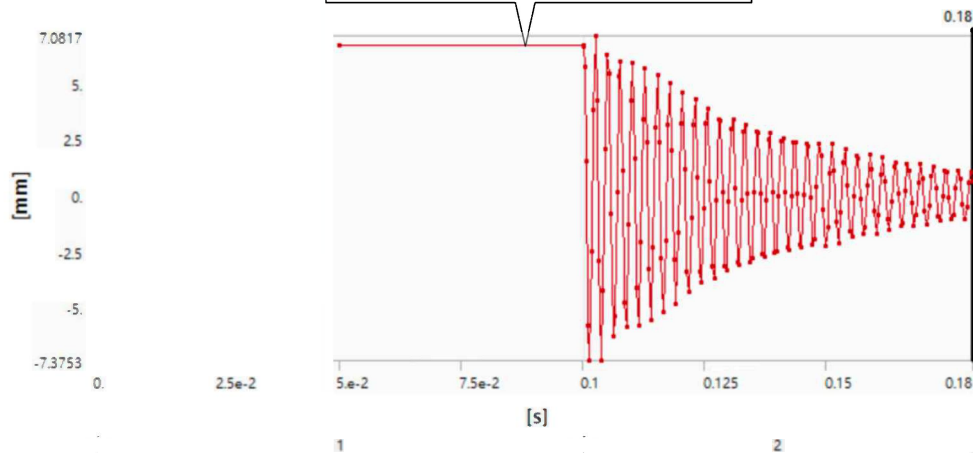
[1] Highlight **Displacement 2** of **Transient** and reselect a quarter point (see [2]). ↓

[2] Now, the displacement is now applied at a quarter point. →

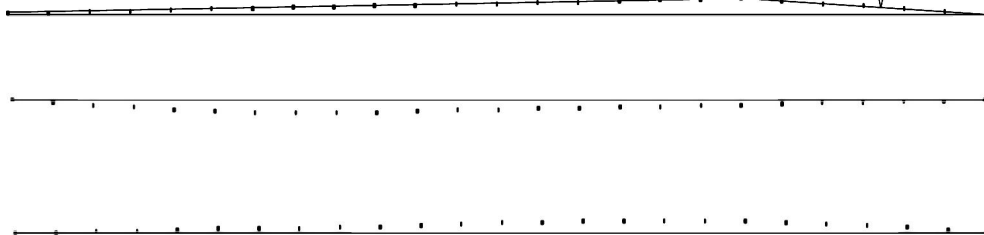


[3] Click **Solve**. ←

[4] The deflection-versus-time plot for the middle point. ↓




[5] Animate the deformation. #

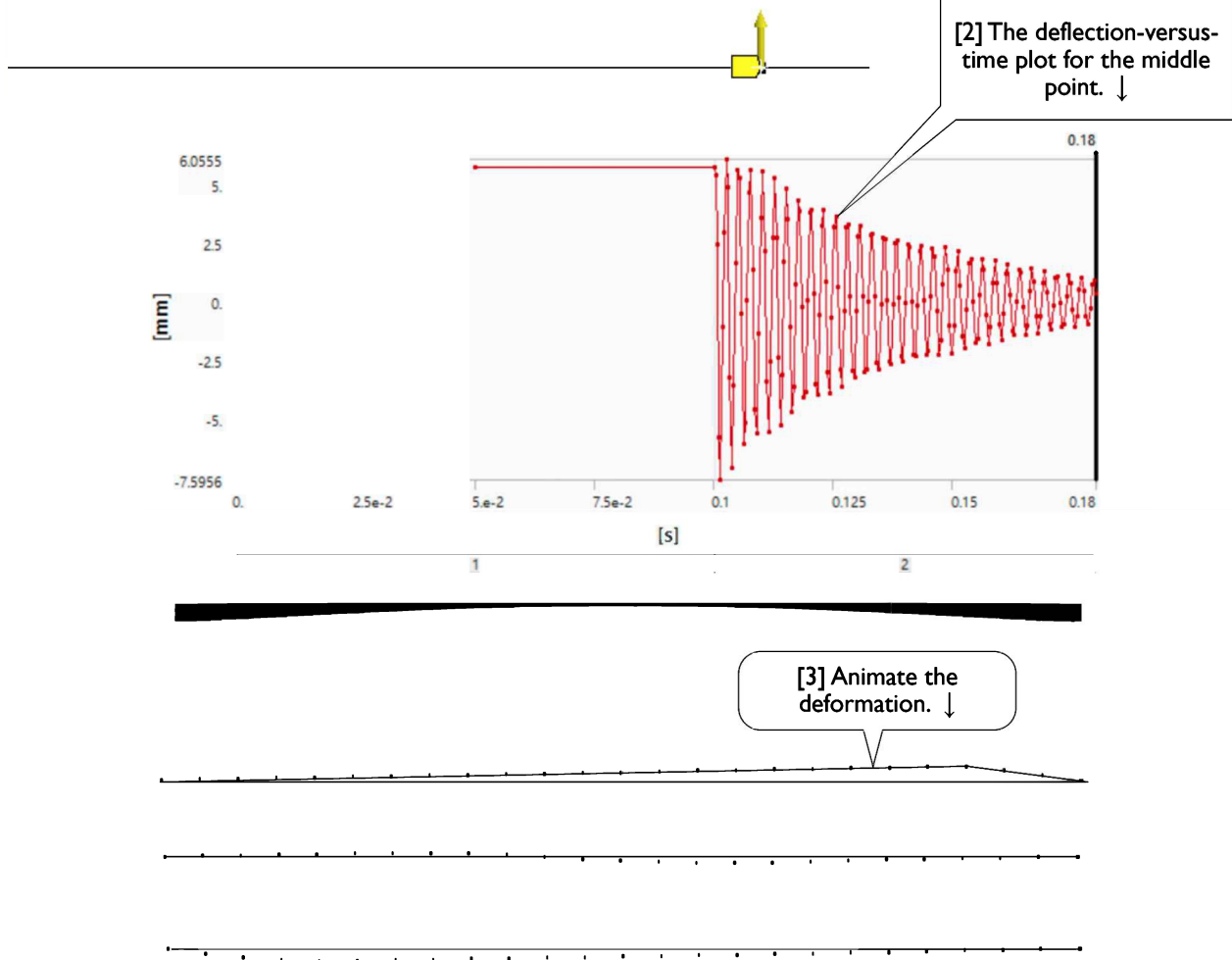


12.5.11 Pluck at Eighth Point

Details of "Displacement 2"	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
[-] Definition	
Type	Displacement
Base Excitation	No
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
Y Component	Tabular Data
Z Component	Free
Suppressed	No

[1] Highlight **Displacement 2**, reselect an eighth point, and solve again. ↓

 Displacement 2
Components: Free, 10, Free mm



Wrap Up

[4] Save the project and exit Workbench. #

Section 12.6

Review

12.6.1 Keywords

Choose a letter for each keyword, from the list of descriptions

- | | |
|-------------------------------------|---|
| 1. () Coulomb Damping | 7. () No Separation Contact |
| 2. () Critical Damping | 8. () Random Vibration Analysis |
| 3. () Explicit Dynamics | 9. () Response Spectrum Analysis |
| 4. () Harmonic Response Analysis | 10. () Steps, Substeps, and Equilibrium Iterations |
| 5. () Lumped Mass Model | 11. () Viscous Damping |
| 6. () Material Damping | |

Answers:

1. (E) 2. (B) 3. (I) 4. (F) 5. (A) 6. (D) 7. (K) 8. (H) 9. (G) 10. (J)
11. (C)

List of Descriptions

(A) Analysis models that assume the mass concentrated at certain locations. The discrete masses are typically connected by springs and dampers. The models are often used to study dynamic systems.

(B) When the damping of a dynamic system is smaller than a critical value, its free vibration is oscillatory. On the other hand, if the damping is larger than the critical value, the motion is not oscillatory.

(C) The damping due to the friction between the structure and its surrounding fluid. The damping force typically assumes to be proportional to the velocity of the structural displacement.

(D) Also called solid damping or elastic hysteresis. The damping due to the internal friction in the material. The behavior is still an open research topic. In the Workbench, assuming small material damping, we may express it by an equivalent viscous damping. Further, the viscous damping coefficient (of this equivalent viscous damping) is assumed to be a linear combination of the stiffness matrix and the mass matrix. The form of linear combination (alpha and beta) is then determined by lab experiments and data fittings.

- (E) Also called Coulomb friction or dry friction. The damping is due to the friction in the connection between structural members. In Workbench, it can be modeled using frictional contact.
- (F) A dynamic analysis to investigate the maximum response of a structure under steady harmonic (sinusoidal) loads.
- (G) A dynamic analysis to evaluate the maximum response of a structure under loading conditions described by a spectrum representing the maximum response at varying frequencies in known directions to a specific time history. The technique is often used to design structures withstanding multiple short-duration loadings, such as earthquakes.
- (H) A dynamic analysis to evaluate the probabilistic response of a structure under probabilistic loads described by a spectrum representing probability distribution of excitation at varying frequencies in known directions. The technique is used to design structures withstanding random loadings.
- (I) A technique for transient dynamic analysis. Explicit integration method is used, which requires a very small integration time step to achieve solution accuracy. For a single time step, it is much more efficient than an implicit method, which is used in a **Transient Structural** analysis system. These features make it very efficient for a high-speed impact simulation, or highly nonlinear simulations.
- (J) We may divide the whole loading history into steps, to specify different **Analysis Settings** for each step. Each step can be further divided into substeps. For transient dynamic simulation, each substep is an integration time step. For nonlinear simulation, dividing into substeps is to expedite convergence. In nonlinear simulations, each substep may need several equilibrium iterations to find its solution.
- (K) Two surfaces with **No Separation** contact condition prohibit separation in their normal direction, but allow small sliding relative to each other. Both **Bonded** and **No Separation** contact types do not introduce contact nonlinearity.

12.6.2. Additional Workbench Exercises

High-Speed Impact Simulation

The impact simulation in the Section 12.4 is actually not useful, since the material (of Young's Modulus 10 kPa) is not realistic and the impact speed (0.5 m/s) is slow. Try more realistic simulations by yourself using the same geometric model. Gradually increase the Young's modulus and the impact speed. Each time you may need to decrease the integration time step. This exercise is to experience the limitation of the implicit method, which is used in a **Transient Structural** analysis system. High-speed impact simulations are more suitable by using **Explicit Dynamics** analysis system (Chapter 15).

Chapter 13

Nonlinear Simulations

When the relationship between the response and the load of a structure is linear, the structure is a *linear structure*, and the simulation is a *linear simulation*. Otherwise, the structure is a *nonlinear structure* and the simulation is a *nonlinear simulation*. In the real world, all structures are more or less nonlinear. In many cases, however, when nonlinearities are negligible, we may predict their behavior using linear simulations. For other cases, when nonlinearities are not negligible, nonlinear simulations are needed.

Structural nonlinearities come from three sources: large deformation, change of connectivity, and nonlinear stress-strain relations. Nonlinearity due to large deformation is called *geometry nonlinearity*. Nonlinearity due to the change of connectivity is called *topology nonlinearity*, which includes failure of structural components and change of contact status. In this chapter, we will discuss the change of contact status only, which can be termed *contact nonlinearity*. Nonlinearity due to nonlinear stress-strain relations is called *material nonlinearity* and will be covered in Chapter 14.

In general, nonlinear simulations are much more challenging than linear simulations. They not only take much computing time, but sometimes fail to find a solution. Solution behaviors of nonlinear simulations highly depend on settings of solution parameters. A thorough comprehension of these solution parameters becomes critical when you want to adjust these parameters to reduce the computing time, or try to successfully find a solution.

Purpose of This Chapter

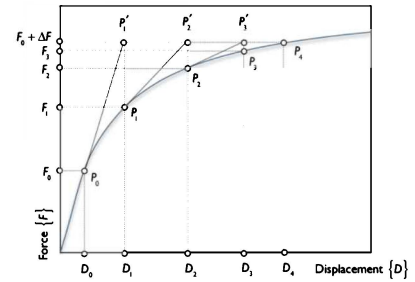
This chapter discusses nonlinear solution algorithms and focuses on geometry nonlinearity and contact nonlinearity. Material nonlinearity will be covered in Chapter 14. This chapter provides some basics of nonlinear simulations process, so that you can understand and use various solution parameters.

About Each Section

Section 13.1 provides some basics of nonlinear simulations, including Newton-Raphson method, the solution method used in Workbench. Concepts of convergence follow the introduction. Section 13.2 provides a step-by-step example of geometric nonlinearity. Section 13.3 provides a step-by-step example of contact nonlinearity. Section 13.4 provides an additional exercise of contact nonlinearity.

Section 13.1

Basics of Nonlinear Simulations



PART A. NONLINEAR SOLUTION METHODS

13.1.1 What Are Nonlinear Simulations

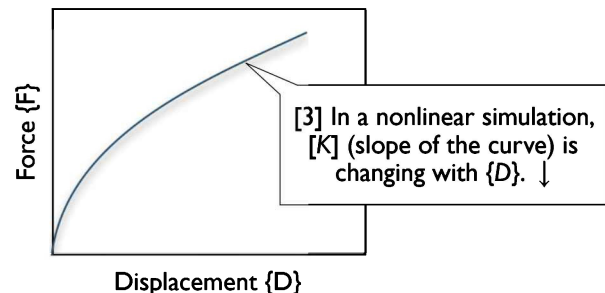
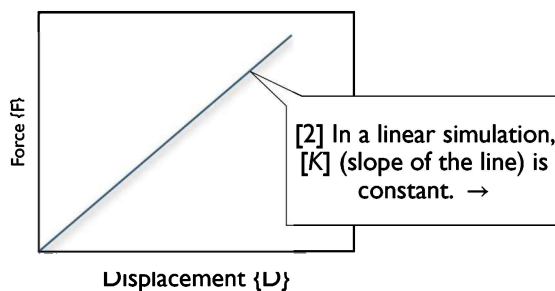
[1] In a linear simulation, the response is linearly proportional to the load. More specifically, let's use Eq. 1.3.1(1) for the upcoming discussion

$$[K]\{D\} = \{F\}$$

Copy of Eq. 1.3.1(1), page 35

The nodal force $\{F\}$ may represent the load; the nodal displacement $\{D\}$ may represent the response (stress and strain can be computed from the displacement); the stiffness matrix $[K]$ contains the proportionality coefficients between the force and the displacement. Eq. 1.3.1(1) can be conceptually plotted as shown in [2]. Note that both the horizontal axis and the vertical axis are actually multi-dimensional (i.e., vectors), and $[K]$ is actually the gradient of $\{F\}$ with respect to $\{D\}$. $[K]$ is a matrix of dimension n by n , where n is the degrees of freedom of the system. In cases of single degree of freedom, $[K]$ is a scalar and is the "spring constant" of the structure.

Performing a linear static simulation means to solve Eq. 1.3.1(1), once and for all. For linear transient dynamic simulations, Eq. 12.1.4(1) (page 426) is solved instead; it involves integrations over time domain. In each integration time step, the equation is solved exactly once. ↓



[4] In a nonlinear simulation, the relation between $\{F\}$ and $\{D\}$ is nonlinear, as shown in [3]. Note that, in [3], we've shown a "concave down" curve, but it may also be a "concave up" curve, or even a curve with inflection points. In nonlinear cases, $[K]$ matrix in Eq. 1.3.1(1) is no longer a constant matrix, it changes with $\{D\}$; that is, $[K]$ is a function of $\{D\}$. To emphasize this, we may rewrite

$$[K(D)]\{D\} = \{F\} \quad (1)$$

Challenges of nonlinear simulations come from the difficulties of solving Eq. (1). #

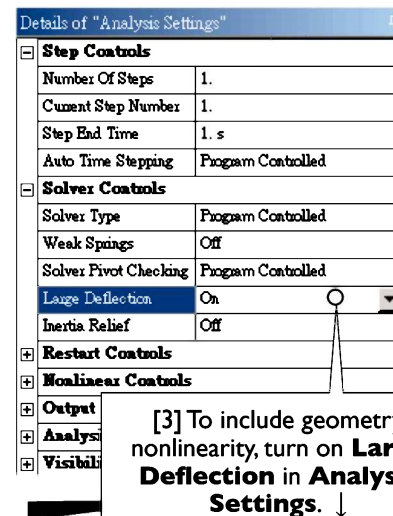
13.1.2 Causes of Structural Nonlinearities

[1] As mentioned in the opening of this chapter, sources of structural nonlinearities can be classified into three categories: geometry nonlinearity, topology nonlinearity, and material nonlinearity. A problem may include more than one category of nonlinearities.

A linear simulation implies that no nonlinearities are present; it in turn implies (a) the deformation is very small, (b) there is no topological change, and (c) the stress-strain relation is linear; i.e., it can be described by Hooke's law. It also implies that the principle of superposition is applicable and the solution is independent of loading history. On the other hand, in nonlinear problems, the principle of superposition is not applicable and the solution may depend on loading history. ↓

Geometry Nonlinearity

[2] Geometry nonlinearity is due to large deformation of structures. The stiffness matrix $[K]$ is composed by element stiffness matrices, and each element stiffness is a function of the element's material properties as well as geometry. When the deformation of structure is so large that the stiffness matrix $[K]$ is changed substantially, geometry nonlinearity must be considered. To include geometry nonlinearity, simply turn on **Large Deflection** in **Analysis Settings** [3]. Among the three sources of nonlinearities, geometry nonlinearity is usually the easiest to tackle: reducing time steps is usually enough to improve the convergence. Some exceptions (e.g., the pneumatic finger of Section 9.1) need special treatments. In this book, so far, we've experienced many simulations involving geometry nonlinearity. Section 13.2 provides one more exercise for geometry nonlinearity. →



Topology Nonlinearity

[4] When the topology (connectivity) of a structure changes, its stiffness matrix also changes. Possible topology changes include failure of structural members or materials, and the changes of contact status. In this chapter, we will cover only contact nonlinearity; i.e., change of contact status.

Contact nonlinearity itself is challenging and, moreover, it is usually accompanied by large deformation. In this book, so far, we've experienced many simulations involving contact nonlinearity. Sections 13.3 and 13.4 are two more exercises for the contact nonlinearity.

Material Nonlinearity

When its material's stress-strain relation is not linear (i.e., it cannot be described by Hooke's law), the problem involves material nonlinearity. In these cases, we need other ways of describing the relation between stress and strain (or stress and strain rate, when dealing with viscous materials). A mathematical model used to describe a stress-strain relationship is called a material model. Eqs. 1.2.8(3-4) (page 32), are two examples of material models. A material model is usually a mathematic form with some parameters; these parameters are usually determined by data fitting using material test data.

Besides Hooke's law, Workbench provides many other material models. Use of these material models is one of the most challenging tasks in the finite element simulations. We will discuss a few material models in Chapter 14. #

13.1.3 Load Steps, Substeps, and Equilibrium Iterations

Steps (Load Steps)

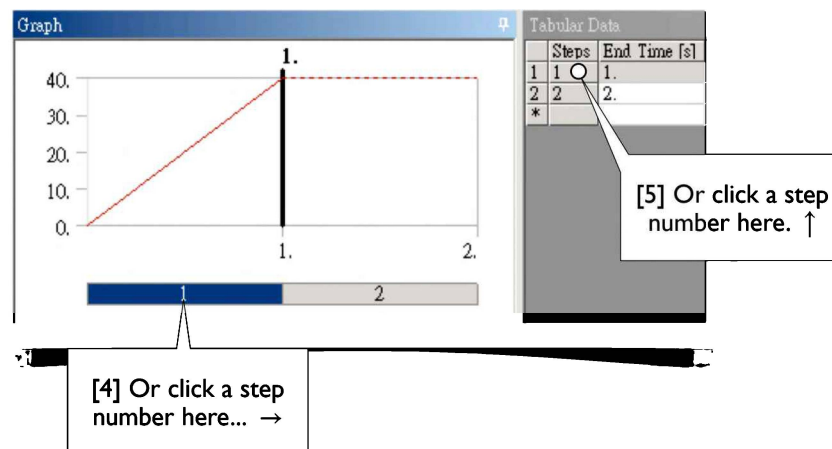
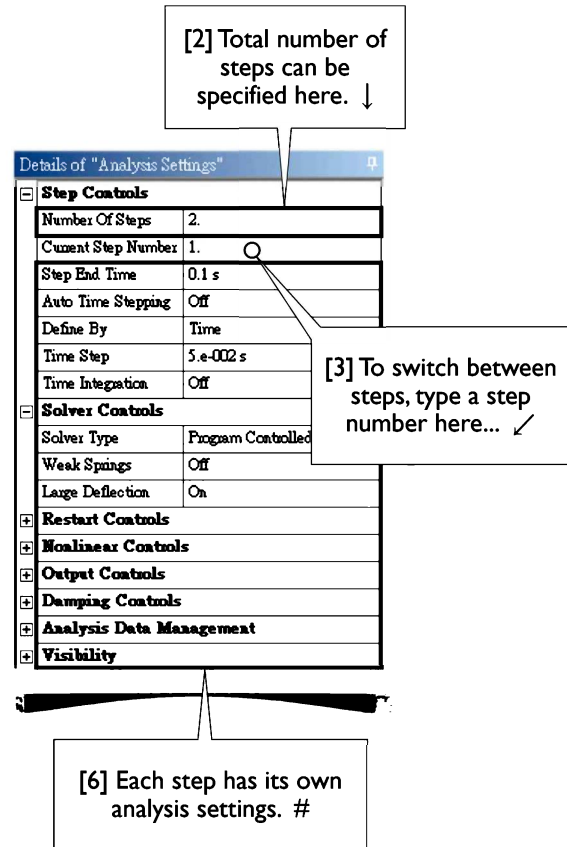
[1] You can divide the entire loading history into one or more *load steps*, or simply called *steps*. The number of steps can be specified in the details view of **Analysis Settings** [2]. To switch between steps, you can type a step number in the details view [3], click a step number in **Graph** [4], or click a step number in **Tabular Data** [5]. Each step can have its own analysis settings [6].

Time Steps (Substeps)

Each load step is further divided into substeps, or time steps. In dynamic simulations, time step is used for integration over time domain (Chapter 12); main consideration of the time step size is to capture the response characteristics. In static simulation, a load step can be divided into substeps to achieve or enhance convergence. Smaller time step size usually converges easier, but, of course, needs a greater number of time steps to complete a load step.

Iterations (Equilibrium Iterations)

For nonlinear problems, each time step itself needs several iterations to solve Eq. 13.1.1(1) (page 469). Each iteration involves solving a subproblem, Eq. 1.3.1(1) (page 35), the linearized equilibrium equation. Solving Eq. 1.3.1(1) is called an equilibrium iteration, or simply iteration. We will introduce the process of equilibrium iterations, known as the Newton-Raphson method, in the next subsection. ↗



13.1.4 Newton-Raphson Method^[Ref 1]

[1] Suppose that the simulation proceeds at a certain time step [2-4], where the displacement is D_0 , the external force is F_0 , and P_0 represents the point at the response curve described by Eq. 13.1.1(1) (page 469). Now, the time is increased one substep further, so the external force is increased to $F_0 + \Delta F$ [5], and we want to find the displacement at next time step [6].

Starting from the point P_0 , Workbench calculates a *tangent stiffness* $[K(D_0)]$, the linearized stiffness, and solves the following equation

$$[K(D_0)]\{\Delta D\} = \{\Delta F\} \quad (1)$$

The displacement D_0 is increased by ΔD and advances to D_1 . Now, in the D - F space, we are at $(D_1, F_0 + \Delta F)$ (i.e., the point P'_1), far from our goal P_4 . To proceed, we need to "drive" the point P'_1 back to the actual curve (i.e., P_1).

Substituting the displacement D_1 into the left-hand side of the governing equation, Eq. 13.1.1(1), we can calculate the actual force F_1 needed for the displacement D_1 ,

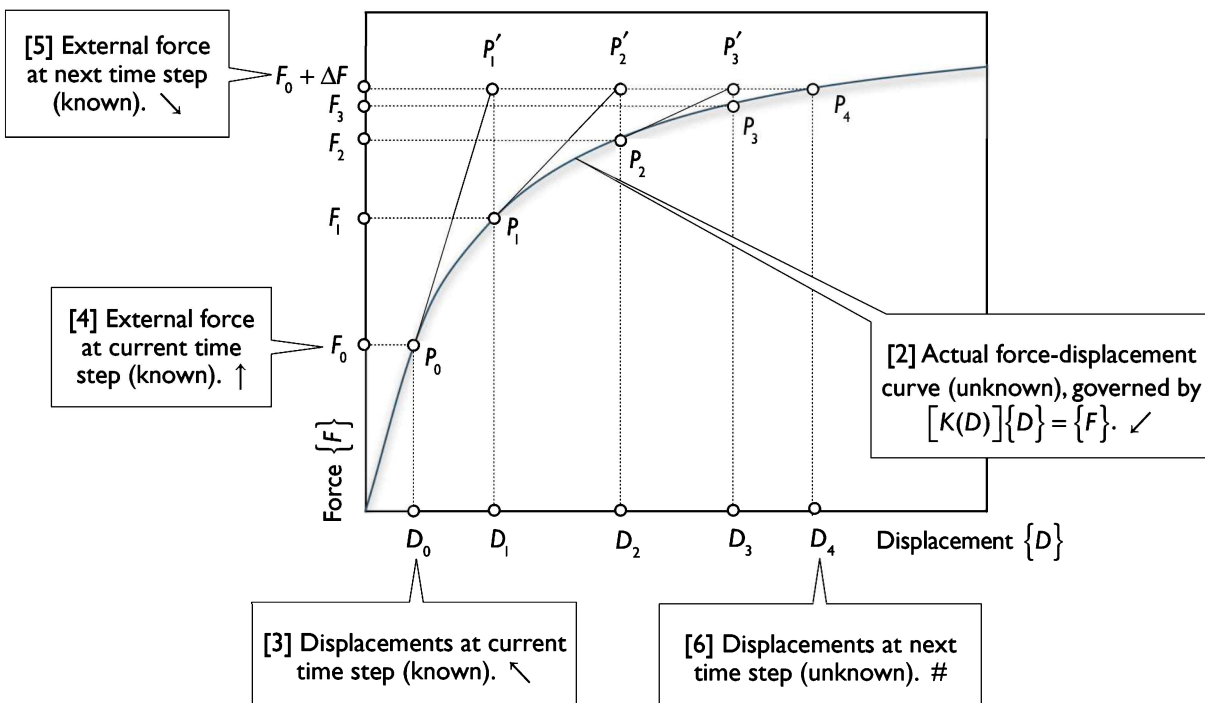
$$[K(D_1)]\{D_1\} = \{F_1\}$$

Now we can locate the point (D_1, F_1) , which is on the actual force-displacement curve. The difference between the external force (here, $F_0 + \Delta F$) and the balanced force (here, F_1) is called the *residual force* of that equilibrium iteration,

$$F_1^R = (F_0 + \Delta F) - F_1$$

If the residual force is smaller than a criterion, then the substep is said to be converged, otherwise, another equilibrium iteration takes place. The iterations repeat until the *convergence criterion* satisfies.

The procedure described above is called the *Newton-Raphson Method*. ↓



13.1.5 Convergence Criteria^[Ref 2]

[1] In the last subsection, we stated that when the residual force F^R is smaller than a criterion, then the substep is converged. This statement is not strictly correct. There are at most four convergence criteria that can be activated under your control, namely, **force convergence** [2], **displacement convergence** [3], **moment convergence** [4], and **rotation convergence** [5]. The moment convergence and rotation convergence can be activated only when shell elements or beam elements are used. These convergence monitoring methods are all defaulted to **Program Controlled**, that is, Workbench automatically turns on any of them when it is appropriate. You may manually turn off or turn on any of them.

When you turn on any of them, you may specify a **Value**, a **Tolerance**, and a **Minimum Reference**. The criterion is then

$$\text{Criterion} = \text{Tolerance} \times \text{maximum}(\text{Value}, \text{Minimum Reference})$$

The force (or moment) convergence satisfies when

$$\|F^R\| < \text{Criterion} \quad (1)$$

The displacement (or rotation) convergence satisfies when

$$\|\Delta D\| < \text{Criterion} \quad (2)$$

where $\|\cdot\|$ denotes the norm of the underlying vector, and is called a **Convergence Value**. Value defaults to ANSYS Calculated, which usually means the current maximum value. In 13.1.4[2-6] (last page), as an example, the current maximum force value is $\|F_0\|$, and the current maximum displacement value is $\|D_0\|$. Tolerances default to 0.5%. Note that setting up a Minimum Reference is to avoid a never-convergent situation when Value is near zero. ↓

[4] When shell elements or beam elements are used, **Moment Convergence** can be activated. ↓

[5] When shell elements or beam elements are used, **Rotation Convergence** can be activated. #

[2] You can turn on **Force Convergence** and set the criterion. ↓

[3] You can turn on **Displacement Convergence** and set the criterion. ←

13.1.6 Solution Information

[1] A text form of convergence values and criteria for each iteration is available in **Solution Information** [2]. A graphics form of this information is also available [3]. A key to tackle nonlinear problems is the ability to interpret and take measures using this information. ↘

Details of "Solution Information"	
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2.5 s
Display Points	All

[2] In **Solver Output**, convergence values and criteria are listed in a text form. ✓

```

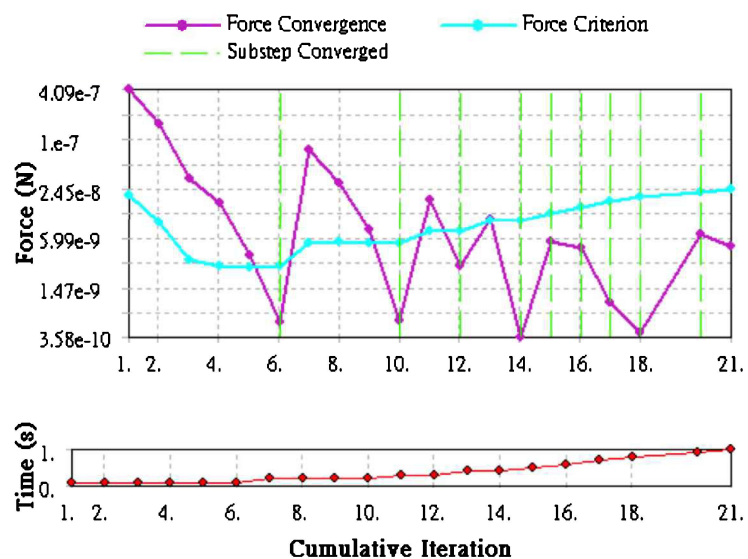
FORCE CONVERGENCE VALUE = 0.9949E-01 CRITERION= 0.2227E-01
EQUIL ITER 1 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.9229E-02
LINE SEARCH PARAMETER = 1.000 SCALED MAX DOF INC = 0.9229E-02
FORCE CONVERGENCE VALUE = 0.6719E-02 CRITERION= 0.2173E-01 <<< CONVERGED
>>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATION 1
*** LOAD STEP 1 SUBSTEP 9 COMPLETED. CUM ITER = 20
*** TIME = 0.900000 TIME INC = 0.100000

FORCE CONVERGENCE VALUE = 0.9542E-01 CRITERION= 0.2455E-01
EQUIL ITER 1 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.9888E-02
LINE SEARCH PARAMETER = 1.000 SCALED MAX DOF INC = 0.9888E-02
FORCE CONVERGENCE VALUE = 0.4753E-02 CRITERION= 0.2415E-01 <<< CONVERGED
>>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATION 1
*** LOAD STEP 1 SUBSTEP 10 COMPLETED. CUM ITER = 21
*** TIME = 1.000000 TIME INC = 0.100000

```

Details of "Solution Information"	
Solution Information	
Solution Output	Force Convergence
Newton-Raphson Residuals	0
Update Interval	2.5 s
Display Points	All

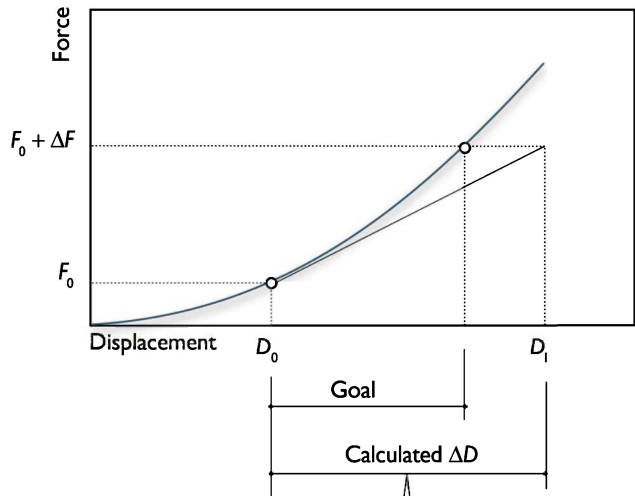
[3] Convergence values and criteria are displayed in graphics form. #



13.1.7 Line Search^[Ref 3]

[1] In the example of 13.1.4[2-6] (page 472), each equilibrium iteration (Eq. 13.1.4(1), page 472) calculates a displacement ΔD , that is smaller than the goal, therefore several iterations are needed to reach the goal. That is true in cases when the F - D curves are monotonically "concave down," such as 13.1.4[2].

In cases when F - D curves are highly nonlinear or "concave up," the calculated displacement ΔD in a single iteration may "overshoot" the goal [2]. In such cases, a numerical technique called **Line Search** can be activated to "scale down" the incremental displacement [3-4]. In these cases, **Line Search** is helpful, but takes extra computing time. →



[2] In cases when the F - D curve is highly nonlinear or concave up, the calculated ΔD in a single iteration may overshoot the goal. ✓

Details of "Analysis Settings"	
Step Controls	
Solver Controls	
Restart Controls	
Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	On
Stabilization	Off
Output Controls	
Analysis Data Management	
Visibility	

[3] **Line Search** can be turned on to scale down the incremental displacement. By default, it is **Program Controlled**. →

[4] **Line Search Parameter** scales down the incremental displacement. #

```

FORCE CONVERGENCE VALUE = 327.2      CRITERION= 0.2962
EQUIL ITER 1 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -0.1956
LINE SEARCH PARAMETER = 0.5779      SCALED MAX DOF INC = -0.1131
FORCE CONVERGENCE VALUE = 179.4      CRITERION= 0.2133
EQUIL ITER 2 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.1267E-01
LINE SEARCH PARAMETER = 1.000      SCALED MAX DOF INC = 0.1267E-01
FORCE CONVERGENCE VALUE = 34.92      CRITERION= 0.2253
EQUIL ITER 3 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.1004
LINE SEARCH PARAMETER = 0.6958      SCALED MAX DOF INC = 0.6985E-01
FORCE CONVERGENCE VALUE = 30.58      CRITERION= 0.2916
EQUIL ITER 4 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.1047E-01
LINE SEARCH PARAMETER = 1.000      SCALED MAX DOF INC = 0.1047E-01
FORCE CONVERGENCE VALUE = 5.867      CRITERION= 0.3076
EQUIL ITER 5 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.1734E-01
LINE SEARCH PARAMETER = 0.8577      SCALED MAX DOF INC = 0.1487E-01
FORCE CONVERGENCE VALUE = 2.724      CRITERION= 0.3313
EQUIL ITER 6 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.3472E-03
LINE SEARCH PARAMETER = 1.000      SCALED MAX DOF INC = 0.3472E-03
FORCE CONVERGENCE VALUE = 0.2257      CRITERION= 0.3382 <<< CONVERGED
>>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATION 6
*** LOAD STEP 2 SUBSTEP 114 COMPLETED. CUM ITER = 1889
*** TIME = 1.89395 TIME INC = 0.288325E-01
*** AUTO TIME STEP: NEXT TIME INC = 0.43249E-01 INCREASED (FACTOR = 1.5000)

```

PART B. CONTACT NONLINEARITY

13.1.8 Contact Types^[Ref 4]

[1] Several contact types are available in Workbench [2]: **Bonded**, **No Separation**, **Frictionless**, **Rough**, **Frictional**, and **Forced Frictional Sliding**, described as follows according to increasing degree of nonlinearity.

Bonded

Two faces (or edges) in **Bonded** contact are coupled together both in their tangential direction and normal direction. No contact nonlinearities are introduced.

No Separation

Two faces (or edges) in **No Separation** contact are coupled in their normal direction only. The tangential direction allows a small sliding on each other. No contact nonlinearities are introduced since small displacement theory is assumed for the sliding.

Rough

Two faces (or edges) in **Rough** contact are free to separate. But, when in contact, they cannot slide in tangential direction, due to large friction between them. This contact type introduces contact nonlinearities. No small displacement theory is assumed.

Frictionless

Two faces (or edges) in **Frictionless** contact are free to separate. And, when in contact, they may slide in tangential direction without any friction force. This contact type introduces contact nonlinearities. No small displacement theory is assumed.

Frictional

Two faces (or edges) in **Frictional** contact are free to separate. And, when in contact, they may slide only when the shear stress between them exceeds a critical value, calculated by multiplying the normal stress by a friction coefficient, which is input as a contact property. This contact type introduces contact nonlinearities. No small displacement theory is assumed.

Forced Frictional Sliding

Similar to **Frictional** except that there is no "sticking" state; i.e., two faces (or edges) slide even when the shear stress is below the critical value. ↗

[2] Several types of contact are available. ↓

Details of "Frictional - Prong To Insert"		4
Scope		
Scoping Method	Geometry Selection	
Contact	4 Edges	
Target	4 Edges	
Contact Bodies	Prong	
Target Bodies	Insert	
Shell Thickness Effect	No	
Definition		
Type	Frictional	
Friction Coefficient	Bonded No Separation Frictionless Rough Frictional Forced Frictional Sliding	
Scope Mode		
Behavior		
Turn Contact		
Suppressed	No	
Advanced		
Formulation	Program Controlled	
Detection Method	Program Controlled	
Penetration Tolerance	Program Controlled	
Elastic Slip Tolerance	Program Controlled	
Normal Stiffness	Program Controlled	
Update Stiffness	Program Controlled	
Stabilization Damping Factor	0	
Pinball Region	Program Controlled	
Time Step Controls	None	
Geometric Modification		
Interface Treatment	Add Offset, No Ramping	
Offset	0. mm	
Contact Geometry Correction	None	

Linear vs. Nonlinear Contacts

[3] The contact types **Bonded** and **No Separation** are called linear contacts because they assume small deformation and involve no nonlinearities. The other contact types will introduce contact nonlinearities. #

13.1.9 Contact versus Target

[1] To specify a contact region, you need to select a set of **Contact** faces (or edges) and a set of **Target** faces (or edges) [2]. During the solution, Workbench checks the contact status for each node (or integration point) on the **Contact** faces against the **Target** faces.

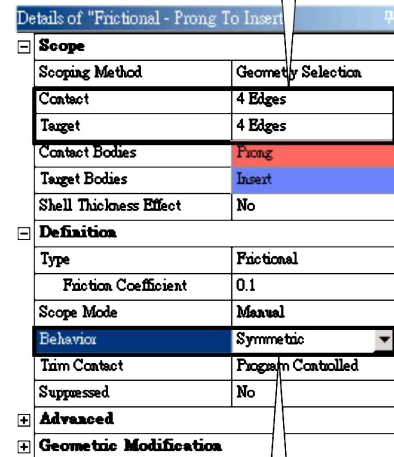
If **Behavior** is set to **Symmetric**, the roles of **Contact** and **Target** will be symmetric [3]; i.e., Workbench checks each point on the **Contact** against the **Target**, as well as each point on the **Target** against the **Contact**. If **Behavior** is set to **Asymmetric**, the checking is only one-sided.

Consider a point, on a **Contact** face, which is approaching a **Target** face. The point is called a contacting point. Workbench keeps tracing the contacting point so that it won't penetrate into the target face. When the point is in contact with the surface, Workbench starts to enforce contact compatibility (i.e., preventing penetration).

In some cases, when **Symmetric** behavior is not available, or when you want to set to **Asymmetric**, to save the run time, selection of **Contact** and **Target** becomes important. You may select **Target** faces using the following guidelines: the faces belong to a body with fixed supports, the faces belong to a body with more rigid material (higher Young's modulus), the faces of less curvature, etc.

For **Symmetric**, results are reported for both contact and target sides. When **Asymmetric** is used, all result data is on the contact side. ↗

[2] To specify a contact region, you need to select a set of **Contact** faces (or edges) and a set of **Target** faces (or edges). ↓



[3] If **Behavior** is set to **Symmetric**, the roles of **Contact** and **Target** will be symmetric. #

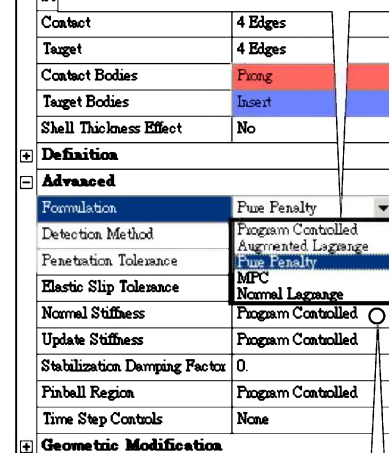
13.1.10 Contact Formulations^[Ref 5]

[1] Workbench offers several **Formulation** options [2] to enforce contact compatibility at the contact interface; i.e., preventing penetration of the contacting point into the target faces.

MPC (Multi-Point Constraint)

For linear contact types (**Bonded** and **No Separation**), a multi-point constraint (**MPC**) formulation is available. **MPC** internally adds constraint equations to couple the displacements between contacting faces. Although you can use other formulations for **Bonded** and **No Separation** contact types, **MPC** is recommended for these linear contact types, since it is a direct, efficient formulation. ↗

[2] Workbench offers several **Formulation** options to enforce contact compatibility. ↓



[3] **Normal Stiffness** (see [4], next page) can be input here. The input value is used to multiply a stiffness value calculated by the program. ↵

Pure Penalty

[4] Whenever a contacting point penetrates normally by an amount x_n into a target face, it will be pushed back by a normal force F_n ,

$$F_n = k_n x_n \quad (1)$$

where k_n is called the **Normal Stiffness** ([3], last page) of the contact region. A **Normal Stiffness** has no real physical meaning; it is a numerical parameter of the penalty algorithm. Solution convergence behavior is usually sensitive to this parameter. A larger k_n usually gives a more accurate solution (less penetration), but may raise convergence issues. Reducing k_n usually helps convergence, but results in increasing penetration. As a simple rule, whenever bumping into a convergence problem, try reducing k_n first. The **Normal Stiffness** can also be automatically adjusted during the solution (see **Update Stiffness** in 13.1.1.1 [1], next page).

If sliding in tangential direction is also prohibited (e.g., **Rough** contact), a similar treatment can be implemented. Whenever a contacting point slides tangentially by an amount x_t , it will be pushed back by a tangential force F_t ,

$$F_t = k_t x_t \quad (2)$$

where k_t is called the *tangential stiffness* of the contact region. Unlike **Normal Stiffness**, the tangential stiffness is always controlled by the program.

Note that, in cases when **Augmented Lagrange** or **Normal Lagrange** (to be discussed) formulation is used, the formulations apply on the normal direction only. The tangential direction still uses **Pure Penalty** formulation, Eq. (2). When **MPC** is used, both normal and tangential directions use **MPC** formulation.

Normal Lagrange

Normal Lagrange formulation adds an extra degree of freedom, namely, contact pressure, to satisfy compatibility. Whenever a contacting point is in touch with the target face, the contact pressure is explicitly calculated. It is the contact pressure that prevents further penetration,

$$F_n = \lambda \quad (3)$$

where λ is the contact pressure, and is traditionally called a *Lagrange multiplier*. Note that this formulation is used in the normal direction only. In the tangential direction, **Pure Penalty** formulation, Eq. (2), is used. This is where the name **Normal Lagrange** comes from.

Normal Lagrange formulation does not require a normal stiffness, and, theoretically, it can enforce zero penetration. However, since no penetration is allowed, the contact status is either open or closed (a step function). This can sometimes make convergence difficult because contact points may oscillate between open and closed status. This behavior is called *chattering*.

Augmented Lagrange

The idea is to combine **Pure Penalty** and **Normal Lagrange**: the push-back normal force is

$$F_n = k_n x_n + \lambda \quad (4)$$

Because of the contact pressure λ , **Augmented Lagrange** formulation is less sensitive to **Normal Stiffness** k_n . Note that again, in the tangential direction, **Pure Penalty** formulation, Eq. (2), is used.

Although **Pure Penalty** is the default setting, **Augmented Lagrange** is recommended for general frictional or frictionless contact in large deformation problems. Since Lagrange method adds an extra degree of freedom, it also takes extra computing time. #

13.1.11 Advanced Contact Settings^[Refs 6, 7]

Pinball Region

[1] The pinball is a sphere region; its radius can be defined in **Pinball Region**. Consider again that a contacting point approaches a target face. If a target node is within the pinball region centered at a contacting node, the contacting node is considered to be in "near" contact with the target node and will be monitored. Target nodes outside of the pinball region will not be monitored.

If **Bonded** type is specified, surfaces that have a gap smaller than the pinball radius are treated as bonded.

Interface Treatment

For **Bonded** contact type, a large enough pinball radius may allow any gap between contacting faces to be ignored. For **Frictional** or **Frictionless** contact types, an initial gap is not automatically ignored, no matter how large the pinball is, since the gap may represent the real geometry.

If an initial gap is present [2] and a force is applied, one part may "fly away" relative to another part [3] if the initial contact is not established right at the end of the time step.

To alleviate situations where a gap (clearance) is modeled but needs to be ignored to establish initial contact for **Frictional** or **Frictionless** contact types, **Interface Treatment** can internally offset the contact surfaces by a specified amount. Note that this treatment is intended for small gaps. Don't apply it in a large gap.

Time Step Controls

Time Step Controls tries to enhance convergence by allowing adjustments of time step size based on contact behavior.

By default, contact behavior does not affect auto time stepping, since adjustment of time step based on contact behavior may increase computing time too much.

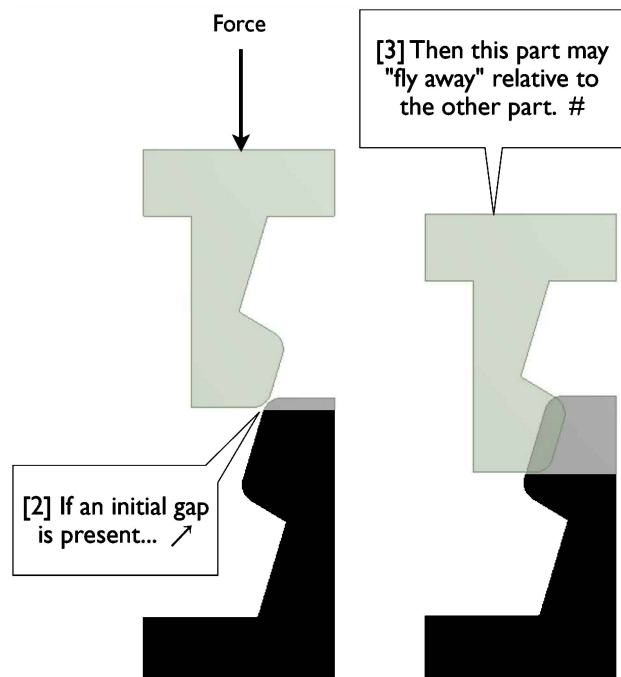
With **Time Step Controls** turned on, Workbench adjusts the time step size based on contact behavior.

Update Stiffness

The **Normal Stiffness** can be automatically adjusted during the solution. Whenever convergence difficulties arise, **Normal Stiffness** will be reduced automatically.

→

Details of "Frictional - Prong To Insert"	
[-] Scope	
Scoping Method	Geometry Selection
Contact	4 Edges
Target	4 Edges
Contact Bodies	Prong
Target Bodies	Insert
Shell Thickness Effect	No
[+] Definition	
[-] Advanced	
Formulation	Pure Penalty
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Each Iteration <input type="radio"/>
Stabilization Damping Factor	0
Pinball Region	Program Controlled <input type="radio"/>
Time Step Controls	None <input type="radio"/>
[-] Geometric Modification	
Interface Treatment	Add Offset, No Ramping <input type="radio"/>
Offset	0. mm <input type="radio"/>
Contact Geometry Correction	None

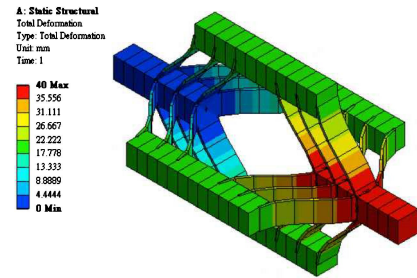


References

1. All Help>Mechanical APDL>Theory Reference>14.11. Newton-Raphson Procedure
2. All Help>Mechanical APDL>Theory Reference>14.11.2. Convergence
3. All Help>Mechanical APDL>Theory Reference>14.11.5. Line Search
4. All Help//Mechanical Application>Mechanical Users' Guide>Setting Connections>Contact>Contact Settings>Definition Settings
5. All Help//Mechanical Application>Mechanical Users' Guide>Setting Connections>Contact>Contact Formulation Theory
6. All Help//Mechanical Application>Mechanical Users' Guide>Setting Connections>Contact>Contact Settings>Advanced Settings
7. All Help//Mechanical Application>Mechanical Users' Guide>Setting Connections>Contact>Contact Settings>Geometric Modification

Section 13.2

Translational Joint^[Ref 1]



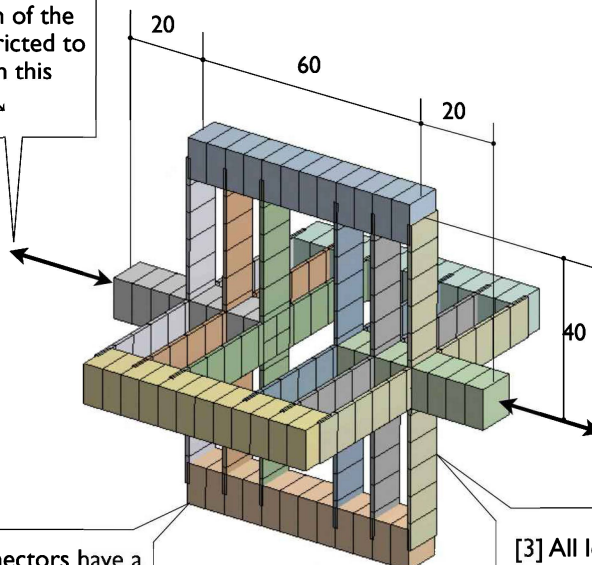
13.2.1 About the Translational Joint

[1] A translational joint is used to connect two machine components, so that the relative motion of the two components is restricted to translation in a specific direction. Conventionally, translational joints are designed as mechanisms, composed by parts, between which the clearance or interference is inevitable; they either decrease precision or increase friction. The translational joint in this section is not a mechanism; rather, it is a unitary flexible structure, in which no clearance or interference exist.

The translational joint [2-5] is made of POM (polyoxymethylene, a plastic), which has a Young's modulus of 2 GPa and a Poisson's ratio of 0.35. The most important design consideration is that the rigidity of translational direction should be much less than all other directions, so that the motion can be restricted in that direction only.

Here, we want to explore the geometric nonlinearity of the structure: how the applied force increases nonlinearly with the translational displacement. For this purpose, we will model the structure using line bodies entirely. The goal of the simulation is to plot a force-versus-displacement chart. The unit system used is **mm-kg-N-s**. ✓

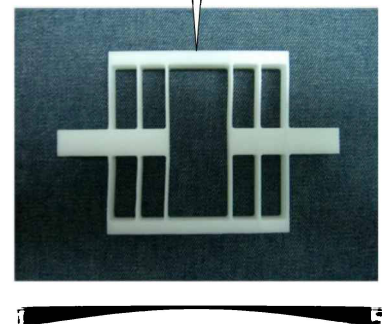
[2] The translational joint is used to connect two machine components, so the relative motion of the components is restricted to translation and in this direction. ↘



[4] All connectors have a cross section of 10 mm x 10 mm. ↗

[3] All leaf springs have a cross section of 1 mm x 10 mm. ←

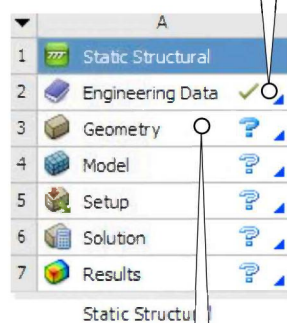
[5] A prototype of translational joint. #



13.2.2 Start Up

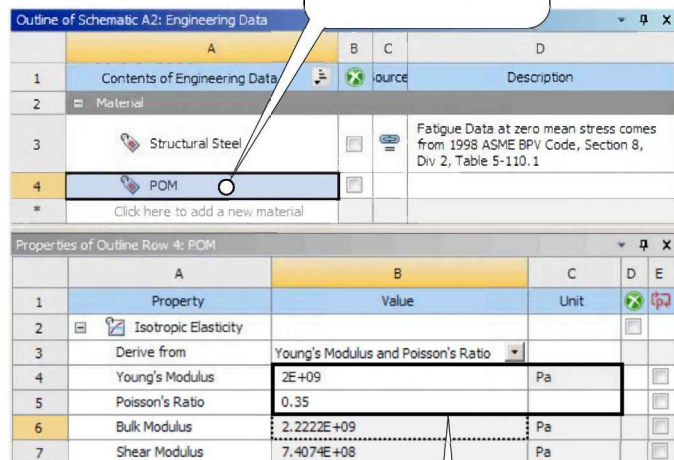
[1] Launch Workbench. Create a **Static Structural** analysis system. Save the project as **Transjoint**. ↓

[2] Double-click **Engineering Data** to prepare the material properties for the POM. →



[5] Start up DesignModeler. Use **Millimeter** as the length unit. #

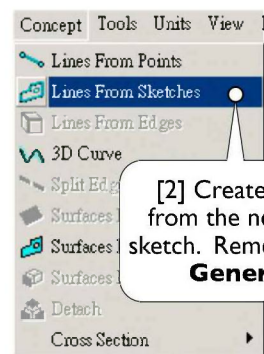
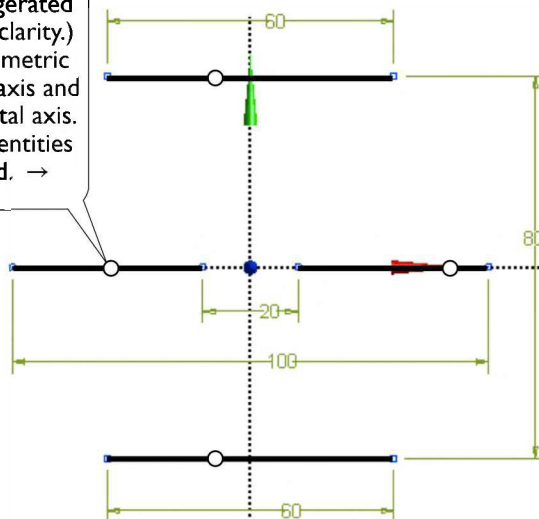
[3] Add a material **POM**. ↓



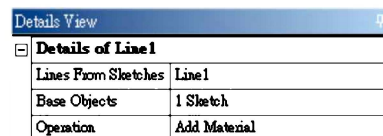
[4] Include **Isotropic Elasticity** and type the properties as shown. Note that SI units are used here. Return to **Project Schematic**. ←

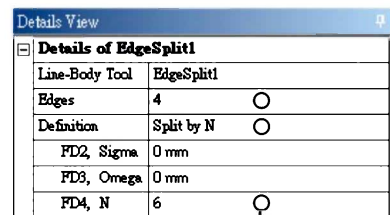
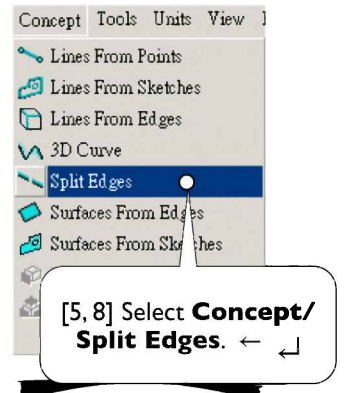
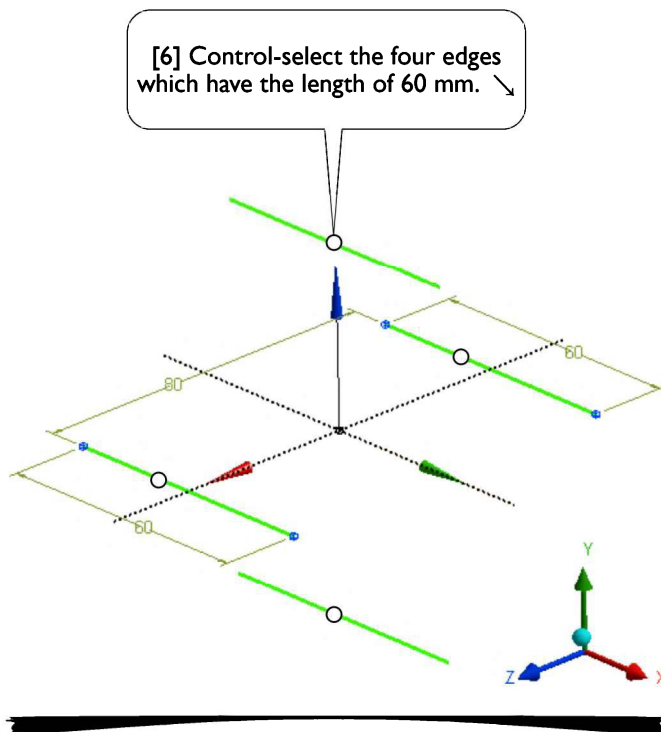
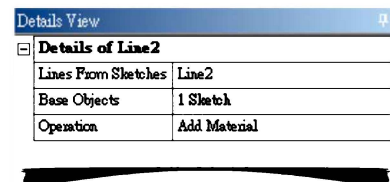
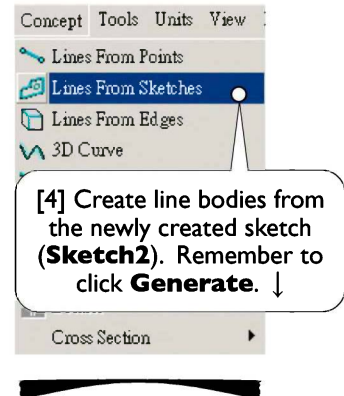
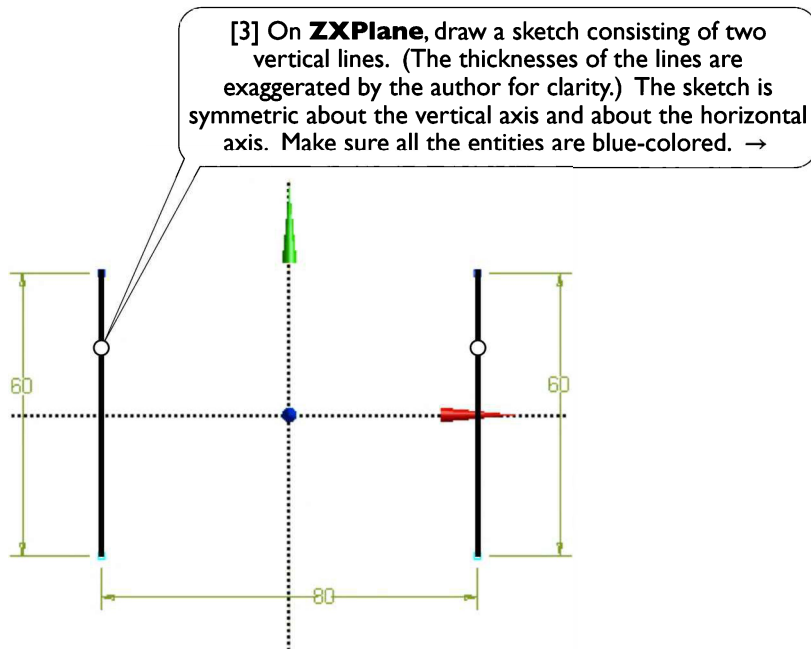
13.2.3 Create Geometry in DesignModeler

[1] On **XYPlane**, draw a sketch of four horizontal lines. (The thicknesses of the lines are exaggerated by the author for clarity.) The sketch is symmetric about the vertical axis and about the horizontal axis. Make sure all the entities are blue-colored. →

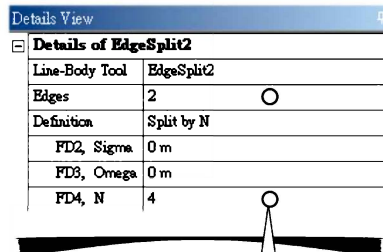


[2] Create line bodies from the newly created sketch. Remember to click **Generate**. ↵



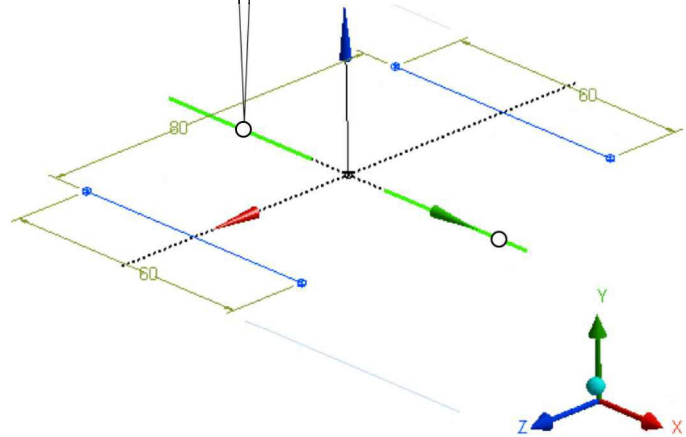


[7] Split each edge into 6 segments. The purpose of the splitting is to create construction points on the edges. Remember to click **Generate**. ↑



[10] Split each edge into 4 segments. Remember to click **Generate**. ↓

[9] Control-select the two edges which have the length of 40 mm. ✓

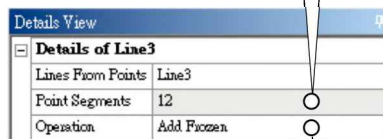


[11] Select **Concept/ Lines From Points**. ↓

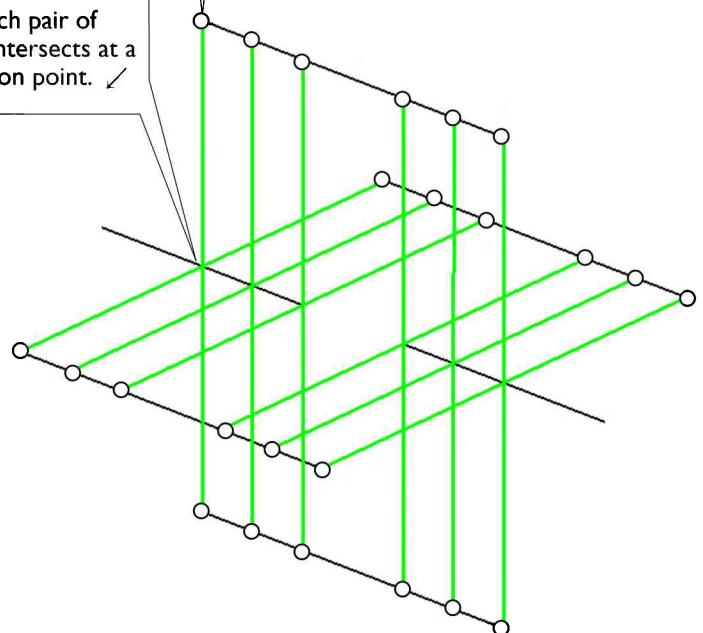
[13] Each segment is created by selecting the starting point then control-selecting the ending point. ✓

[14] Each pair of segments intersects at a construction point. ✓

[12] Create 12 line segments as the leaf springs, as shown in [13, 14]. ↗

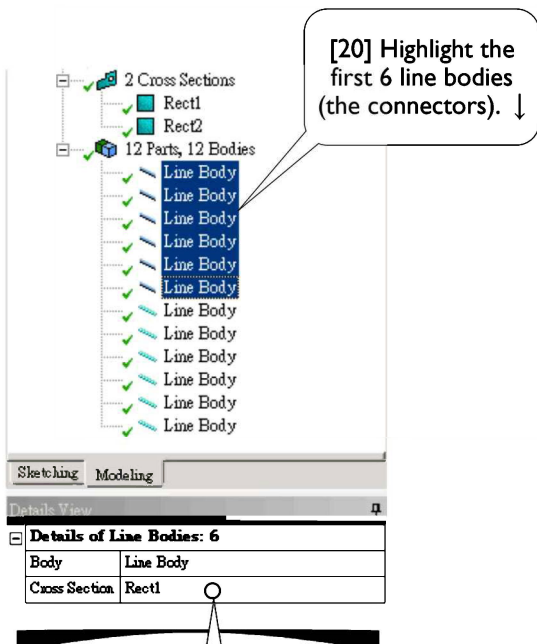
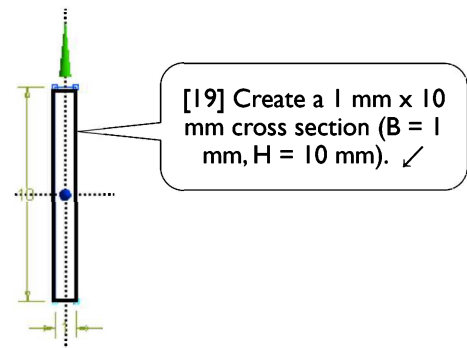
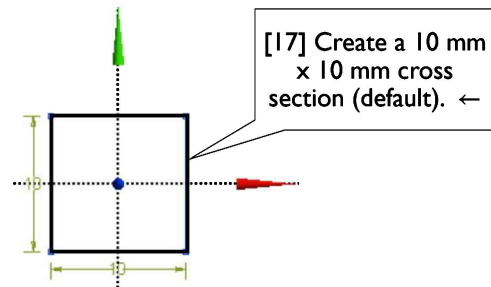
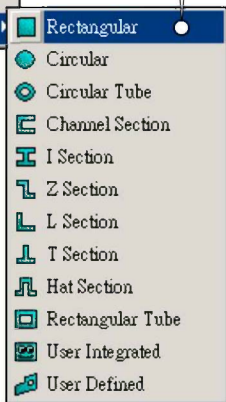


[15] Add these segments as separate parts, since these leaf springs have a different cross section from the other line bodies. Click **Generate**. ↵

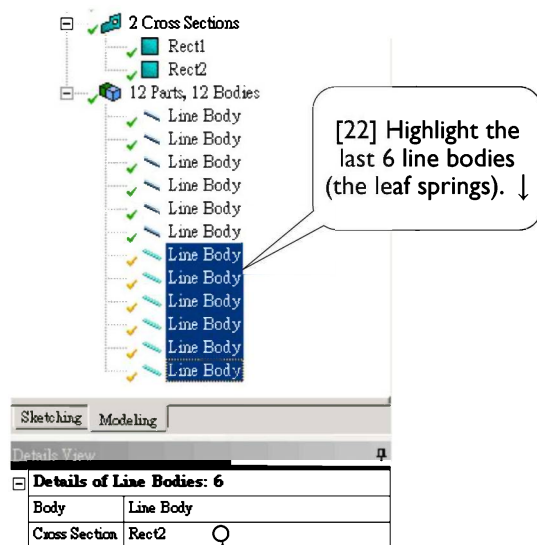




[16, 18] Select **Concept/Cross Section/Rectangular**. → ↘

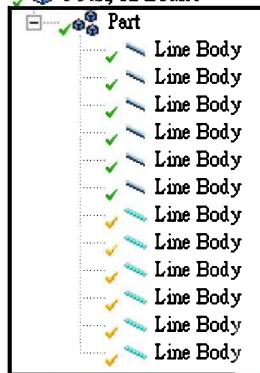


[21] Assign **Rect1**, which has a cross section of 10 mm x 10 mm, to the connectors. ↗

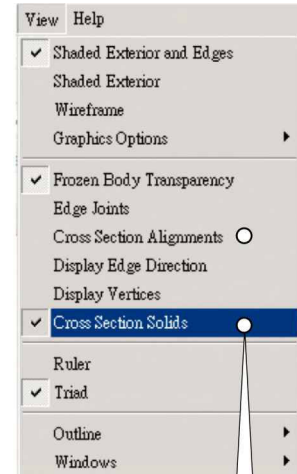
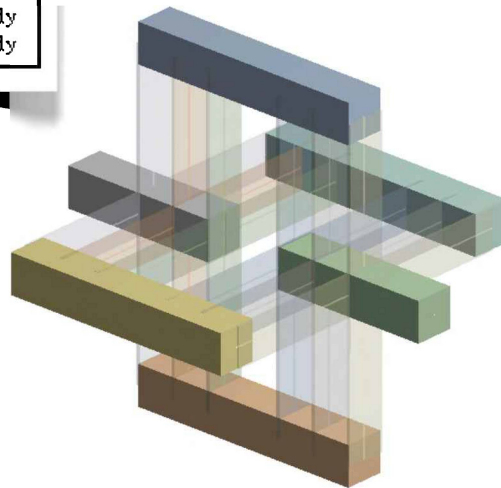


[23] Assign **Rect2**, which has a cross section of 1 mm x 10 mm, to the leaf springs. ↖

1 Part, 12 Bodies

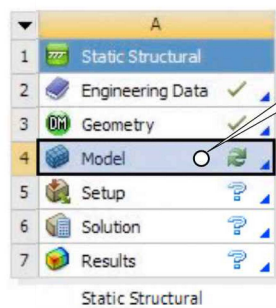


[24] Combine all the line bodies to form a single part. (Select all 12 line bodies and right-click-select **Form New Part**.) ↘



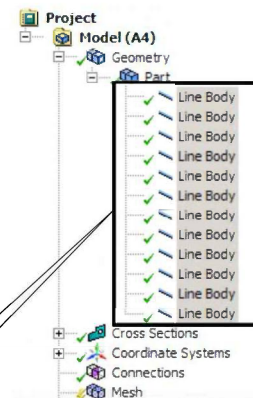
[25] Turn on **View/Cross Section Solids**. Turn off **Cross Section Alignments**. Close DesignModeler. #

13.2.4 Linear Simulation



[1] Start up **Mechanical**. Select the **mm-kg-N-s** unit system. ↓

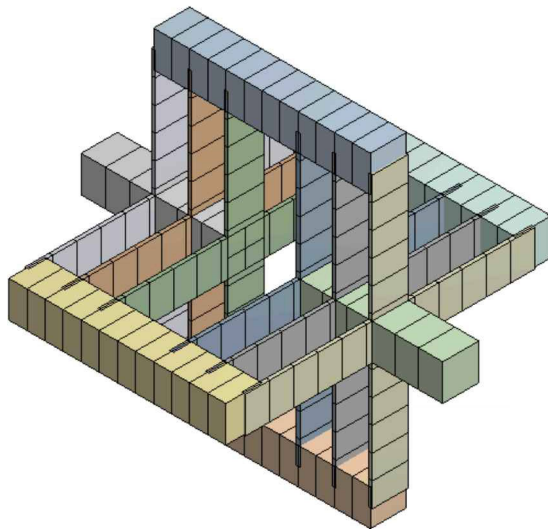
[2] Select all the bodies and assign them the material **POM**. ↓



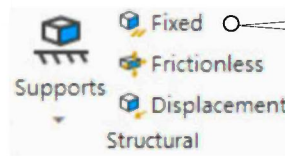
Details of "Multiple Selection"	
Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Cross Section	
Offset Mode	Refresh on Update
Offset Type	Centroid
Model Type	Beam
Material	
Assignment	POM
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	
Statistics	



[3] Generate **Mesh**. ↙

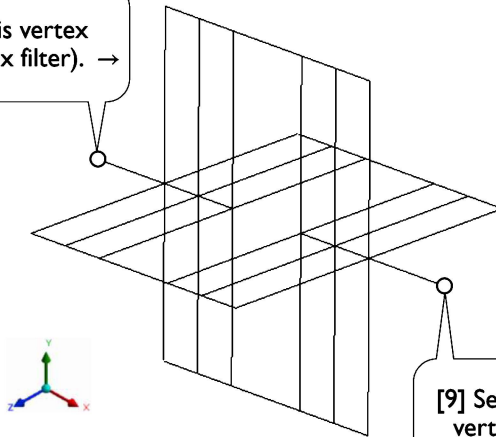


[4] Turn on **Display/Thick Shells and Beams**. The default settings mesh the model with 208 beam elements. That is fine enough for a model like this (see 7.2.8[4], page 289). ↓



[5] With **Static Structural** highlighted, insert a **Fixed Support**. ←

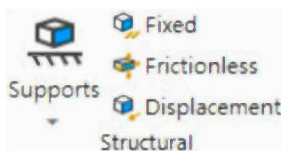
[6] Select this vertex (using the vertex filter). →



[9] Select this vertex. ↘

[7] Click **Apply**. ✓

Details of "Fixed Support"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Fixed Support
Suppressed	No



[8] Insert a **Displacement**. ↑

Details of "Displacement"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	40. mm (ramped)
Y Component	Free
Z Component	Free
Suppressed	No

[10] Click **Apply**. ↓

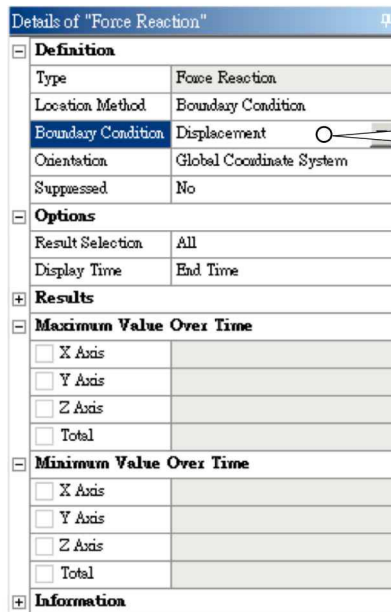
[11] Type 40 (mm) for **X Component**. ↙



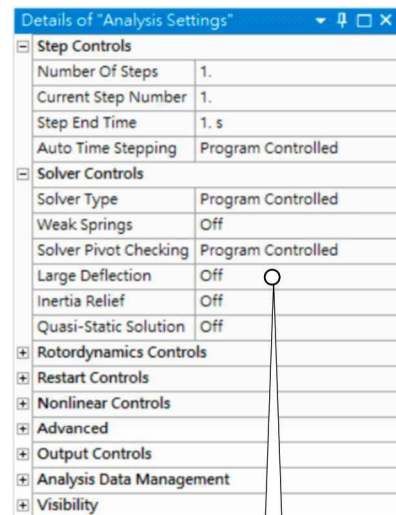
[12] With **Solution** highlighted, insert a **Total Deformation**. →



[13] Insert a **Probe/Force Reaction**. ✓



[14] Select **Displacement**. This evaluates the force required for the 40-mm displacement ([10-11], last page). ↓



[16] The default **Analysis Settings** are like this. Note that **Large Deflection** is off. We now obtain a linear solution first and then obtain a nonlinear solution later. ↓

Force Reaction

[15] Forces are needed to maintain a displacement condition. For example, if a point of a structure is to be held still (i.e., a fixed support), forces are needed to maintain the zero-displacement condition. The forces are called *force reactions*.

This concept can be generalized to a nonzero displacement condition. In our case, we want to maintain a displacement of 40 mm in X-direction; the force required is called the force reaction for that displacement.

The **Force Reaction** object is to evaluate the force required for the 40-mm displacement. →



[17] Click **Solve**. ↩

Details of "Force Reaction"

Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Displacement
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	All
Display Time	End Time
Results	
X Axis	74.67 N
Y Axis	0. N
Z Axis	0. N
Total	74.67 N
+ Maximum Value Over Time	
+ Minimum Value Over Time	
+ Information	

[18] Highlight **Force Reaction**. ↓

[19] The force required to displace 40 mm is 74.67 N. #

13.2.5 Nonlinear Simulation

Details of "Analysis Settings"

Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Quasi-Static Solution	Off
+ Rotordynamics Controls	
+ Restart Controls	
+ Nonlinear Controls	
+ Advanced	
+ Output Controls	
+ Analysis Data Management	
+ Visibility	

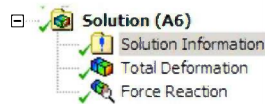
[1] Turn on **Large Deflection**. →

[2] Click **Solve**. ✓

[3] The force required is 101.43 N, which is 36% more than the force obtained in the linear simulation. This justifies the need for a nonlinear simulation. ↵

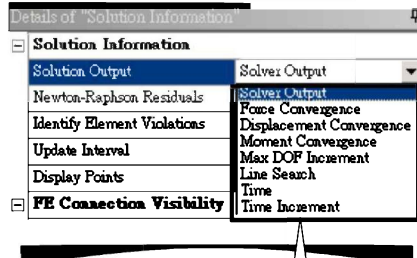
Details of "Force Reaction"

Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Displacement
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	All
Display Time	End Time
Results	
X Axis	101.43 N
Y Axis	0. N
Z Axis	0. N
Total	101.43 N
+ Maximum Value Over Time	
+ Minimum Value Over Time	
+ Information	



[4] Highlight **Solution Information**.

[6] **Solver Output** reports convergence values and criteria in each equilibrium iteration.

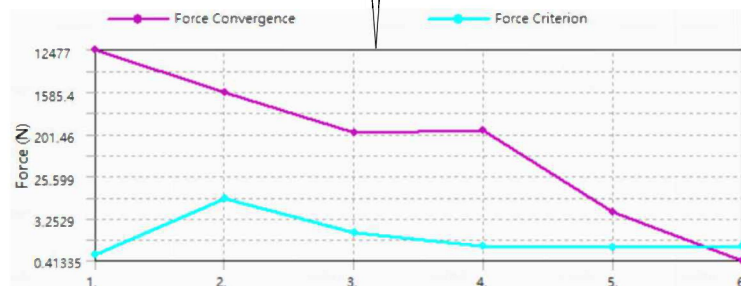


[5] Much information is recorded. Try these options one after another [6-8].

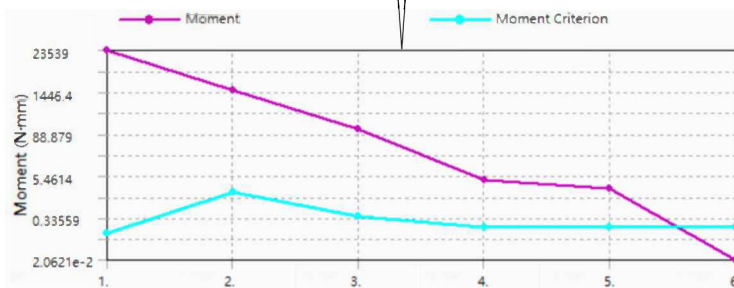
```

DISP CONVERGENCE VALUE = 40.00 CRITERION= 2.041
EQUIL ITER 1 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 40.00
FORCE CONVERGENCE VALUE = 0.1248E+05 CRITERION= 0.5495
MOMENT CONVERGENCE VALUE = 0.2354E+05 CRITERION= 0.1244
DISP CONVERGENCE VALUE = 5.362 CRITERION= 2.082
EQUIL ITER 2 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -5.362
FORCE CONVERGENCE VALUE = 1516. CRITERION= 8.582
MOMENT CONVERGENCE VALUE = 1669. CRITERION= 1.944
DISP CONVERGENCE VALUE = 0.8783 CRITERION= 2.125 <<< CONVERGED
EQUIL ITER 3 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.8725
FORCE CONVERGENCE VALUE = 219.9 CRITERION= 1.642
MOMENT CONVERGENCE VALUE = 120.8 CRITERION= 0.3718
DISP CONVERGENCE VALUE = 0.4238 CRITERION= 2.168 <<< CONVERGED
EQUIL ITER 4 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -0.4081
FORCE CONVERGENCE VALUE = 239.8 CRITERION= 0.8314
MOMENT CONVERGENCE VALUE = 4.287 CRITERION= 0.1883
DISP CONVERGENCE VALUE = 0.8452E-01 CRITERION= 2.213 <<< CONVERGED
EQUIL ITER 5 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.7990E-01
FORCE CONVERGENCE VALUE = 4.454 CRITERION= 0.8178
MOMENT CONVERGENCE VALUE = 2.404 CRITERION= 0.1852
DISP CONVERGENCE VALUE = 0.1539E-01 CRITERION= 2.258 <<< CONVERGED
EQUIL ITER 6 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -0.1509E-01
FORCE CONVERGENCE VALUE = 0.4133 CRITERION= 0.8262 <<< CONVERGED
MOMENT CONVERGENCE VALUE = 0.2062E-01 CRITERION= 0.1871 <<< CONVERGED
>>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATION 6
  
```

[7] **Force Convergence** is a graphics plotting of force convergence values and criteria.

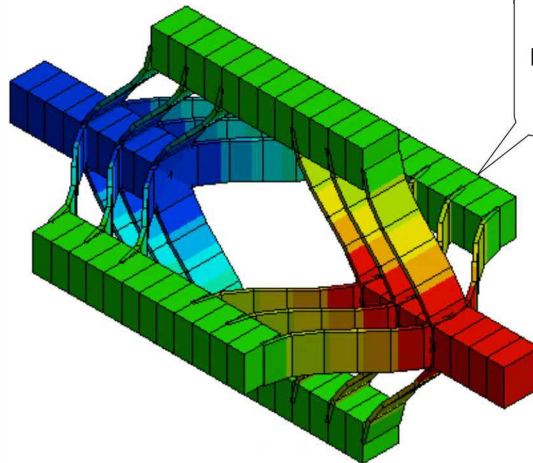


[8] **Moment Convergence** is a graphics plotting of moment convergence values and criteria. Note that **Moment Convergence** is available only for the model with rotational degrees of freedom (i.e., shell or beam elements).



A: Static Structural
 Total Deformation
 Type: Total Deformation
 Unit: mm
 Time: 1

40 Max
 35.556
 31.111
 26.667
 22.222
 17.778
 13.333
 8.8889
 4.4444
0 Min



[10] Workbench solves this case easily in only one substep, and the substep takes 6 equilibrium iterations to converge (see [7-8], last page). To plot a force-versus-displacement chart, let's use a smaller initial substep value and let Workbench automatically adjust the substep values for the subsequent substeps. This feature is called **Auto Time Stepping**. ↓

Details of "Analysis Settings"	
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	On <input checked="" type="radio"/>
Define By	Time <input type="radio"/>
Initial Time Step	0.1 s <input type="radio"/>
Minimum Time Step	1.e-002 s <input type="radio"/>
Maximum Time Step	1. s <input type="radio"/>
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Restart Controls	
Nonlinear Controls	
Output Controls	
Analysis Data Management	
Visibility	

[11] Turn on **Auto Time Stepping**. ↓

[12] Type 0.1 (sec) for **Initial Time Step**. ↓

[13] Set **Minimum Time Step** to 1/10 of the initial time step. ↓

[14] Set **Maximum Time Step** to 10 times the initial time step. →

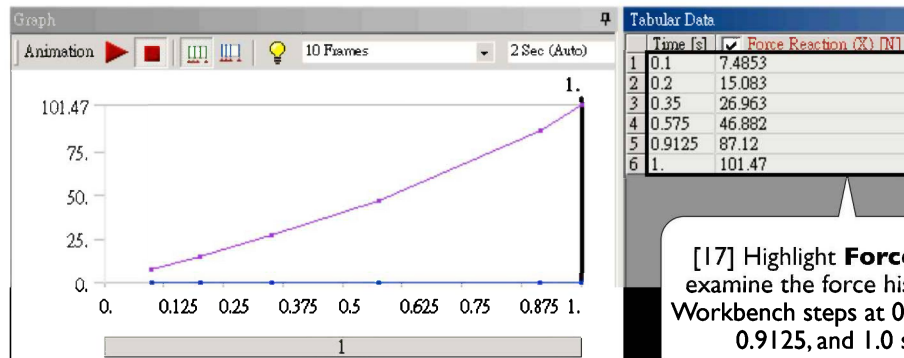
Auto Time Stepping

[15] Workbench uses **Initial Time Step** for the first substep. If the convergence behavior is merely okay, it continues to use that time step for the next substep. If the convergence behavior is very good, it may increase the time step (typically 50%) for the next substep. If the convergence behavior is pretty bad, it may decrease the time step (typically 50%; called *bisection*) for the next substep.

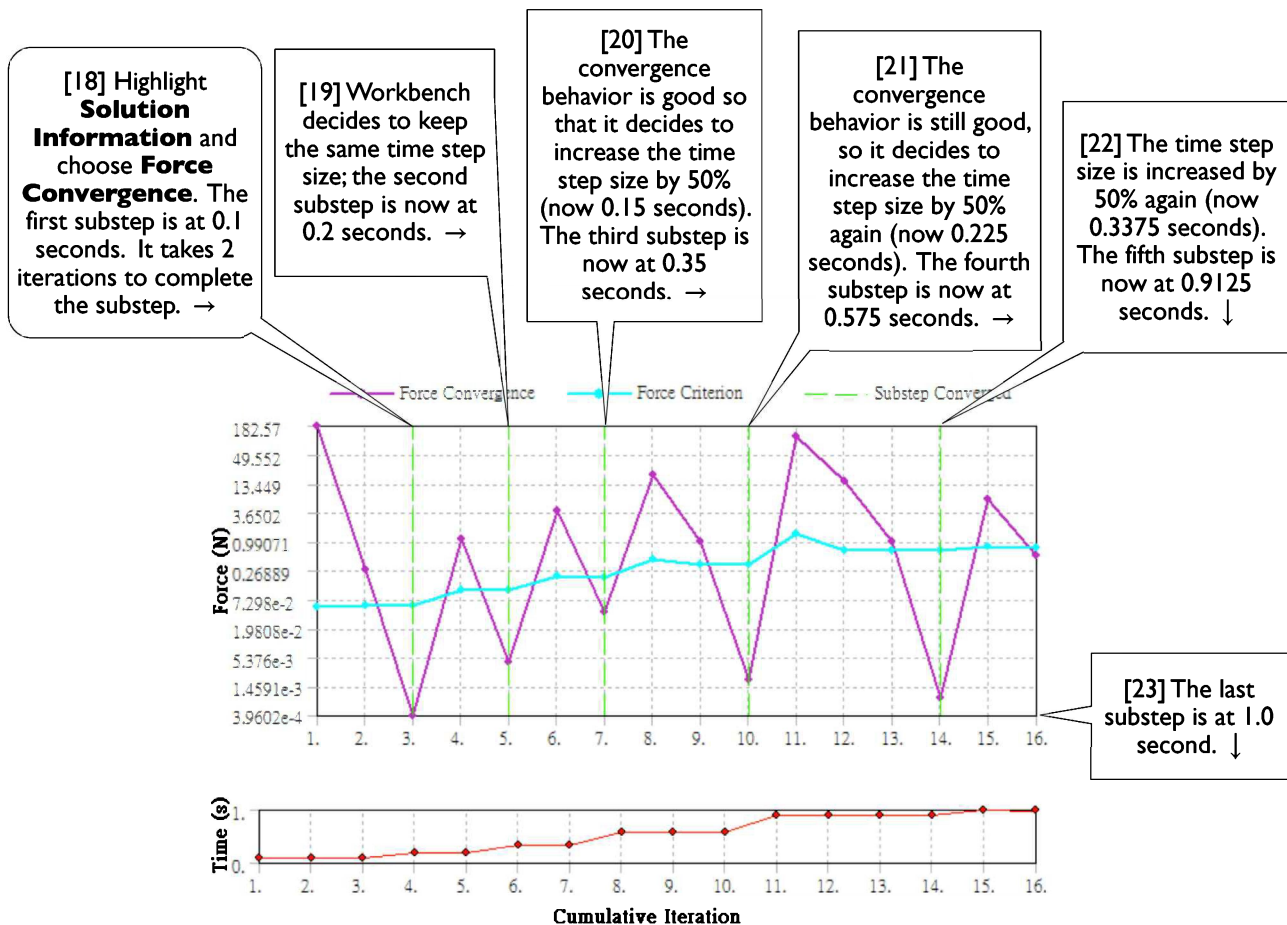
In cases that the convergence behavior is so bad that, after several times of bisections, the time step becomes less than **Minimum Time Step**, Workbench will stop the solution procedure and report an error message for you. ←



[16] Click **Solve**. ←



[17] Highlight **Force Reaction** to examine the force history. Note that Workbench steps at 0.1, 0.2, 0.35, 0.575, 0.9125, and 1.0 seconds. ←



Time is used as a counter in static simulations

[24] In a static simulation, in which the time has no real-world meaning, Workbench uses **Time** as a counter. By default, it set 1.0 second for each step. You, of course, can change that value as you wish. In our case, since the total displacement is 40 mm, we may set **Step End Time** to 40 seconds, so that each second corresponds to a millimeter of displacement. ↵

More data points; more uniform data points

[25] In the foregoing simulation, there are only 6 data points calculated ([17], last page), and they are distributed unevenly. To plot a more accurate force-versus-displacement relation, we need more time steps which preferably distribute evenly. Let's modify **Analysis Settings** further. ↓

Details of "Analysis Settings"

Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	40. s
Auto Time Stepping	Off
Define By	Time
Time Step	2. s
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Restart Controls	
Nonlinear Controls	
Output Controls	
Analysis Data Management	
Visibility	

[26] Set **Step End Time** to 40 (s), so that each second corresponds to a millimeter of displacement. ↓

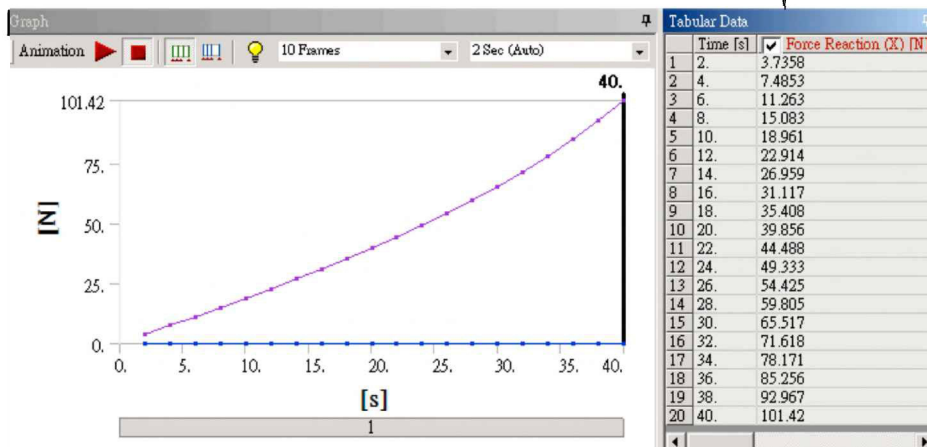
[27] Turn off **Auto Time Stepping**. ↓

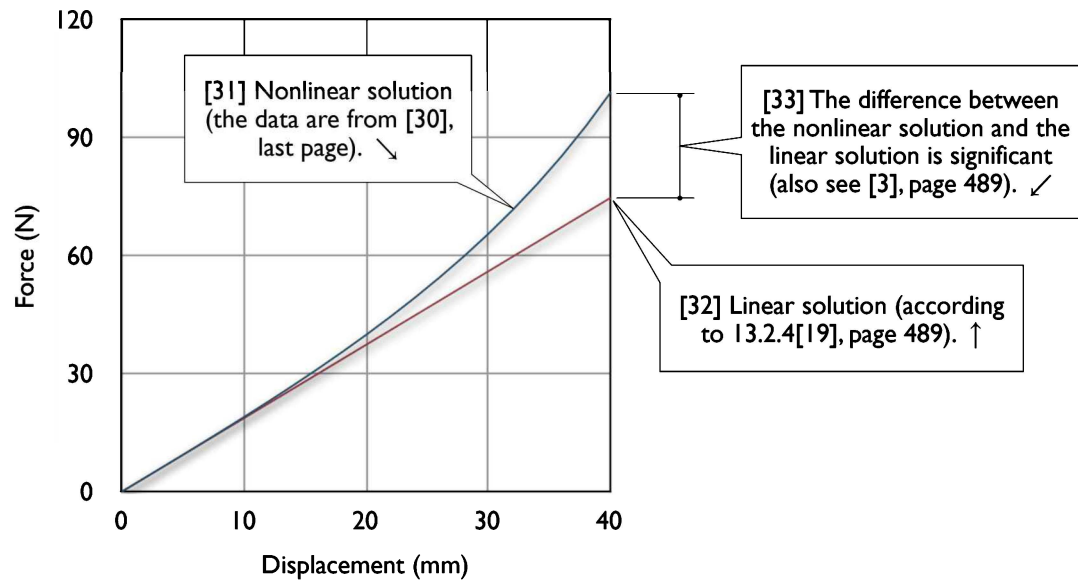
[28] Each **Time Step** is set to 2 seconds, i.e., 2 mm of displacement. →



[29] Click **Solve**. ↓

[30] Highlight **Force Reaction** to examine the force history. ↵





Wrap Up

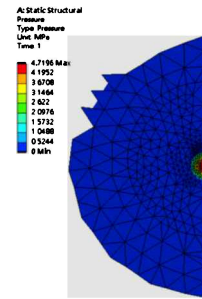
[34] Save the project and exit Workbench. #

Reference

1. Brian P. Trease, Yong-Mo Moon, and Sridhar Kota, 2005, "Design of Large-Displacement Compliant Joints," *ASME Journal of Mechanical Design*, Vol. 127, pp. 788-798.

Section 13.3

Microgripper^[Refs 1,2]



13.3.1 About the Microgripper

[1] In Section 2.6, we introduced the microgripper and created a solid model for it. The microgripper is made of a rubber-like polymer material and actuated by a shape memory alloy (SMA) actuator; it is tested by gripping a steel bead in a lab. In this section, we want to assess the gripping pressure on the bead under an actuation force of $40 \mu\text{N}$ exerted by the SMA. The polymer material has a Young's modulus of 200 MPa and a poisson's ration of 0.48. Since the steel bead is much more rigid than the polymer material, we will model the steel bead as a rigid body. This will ease some of the computation (convergence) difficulties. By considering the symmetries, we will model only one quarter of the microgripper. #

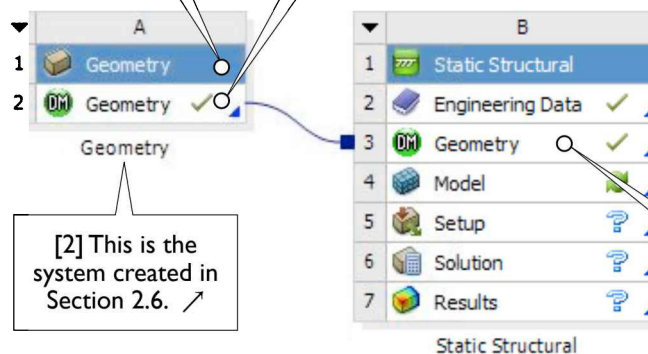
13.3.2 Resume the Project **Microgripper**

[1] Launch Workbench. Open the project **Microgripper**, which was saved in Section 2.6. ↓

[5] We don't need this system any more. Right-click here and select **Delete**. ↘

[3] From **Toolbox**, drag **Static Structural** and drop here. ↓

[2] This is the system created in Section 2.6. ↗



[6] Double-click **Geometry** to start up DesignModeler. #

[4] A **Static Structural** system is created. ↖

13.3.3 Prepare Geometry in DesignModeler

[1] Click **New Plane**. ↓

[2] The new plane is simply a copy of **XYPlane** with its Z-axis reversed. Click **Generate**. →

[3] Select **Tools/Symmetry**. ←

[4] Select **2** for **Number of Planes**. Select the newly created plane (**Plane4**) and **YZPlane** as the planes of symmetry. Click **Generate**. ↓

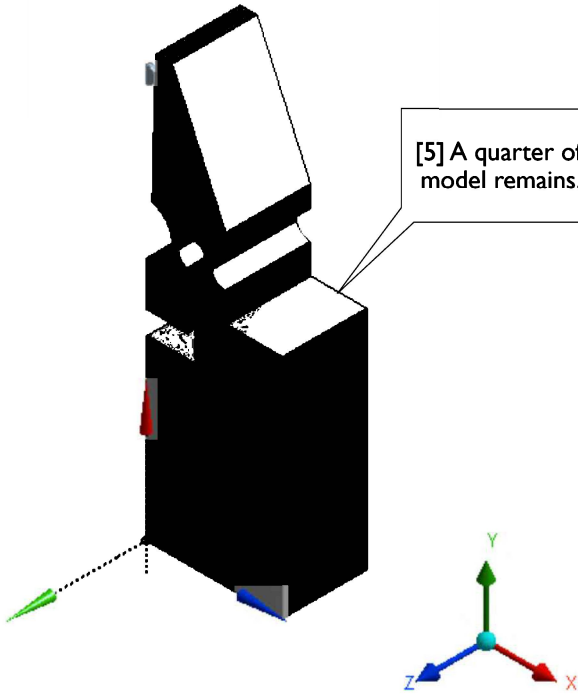
[5] A quarter of the model remains. →

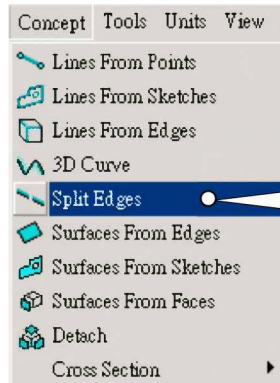
Why use Plane4 instead of XYPlane?

[6] Because we want to keep the half of the model behind **XYPlane**, so that it has better visual effect. When you specify a plane of symmetry, DesignModeler keeps the portion that is in the +Z side of the plane coordinate system.

You don't have to build a full model

To create a **Tools/Symmetry** feature, you don't have to build a full model and then slice it in half. You may just build a half model and use **Tools/Symmetry** to specify a plane of symmetry. The information that Workbench needs is the planes of symmetry, not the other half of the model. ↵





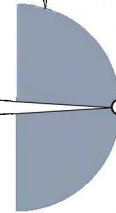
[7] Select **Concept/ Split Edges**. →

[9] Click **Apply**. Remember to click **Generate**. →

Details View		
Details of EdgeSplit1		
Line-Body Tool	EdgeSplit1	
Edges	1	<input type="radio"/>
Definition	Split by N	<input type="radio"/>
FD2, Sigma	0 μm	
FD3, Omega	0 μm	
FD4, N	2	<input type="radio"/>

[8] Rotate to look at **XYPlane** and select this edge (the half circle). ✓

[10] The purpose of splitting the edge is to create a point at the contact point. We will use this point as the center to create imprinting arcs on the nearby faces. ✓



[12] Select **Imprint Faces**. And select the contact point shown in [10] as **Origin Selection**. Click **Generate**. ←

Details View		
Details of Sphere1		
Sphere	Sphere1	
Base Plane	XYPlane	
Operation	Imprint Faces	<input type="radio"/>
Target Bodies	All Bodies	
Origin Definition	Selection	<input type="radio"/>
Origin Selection	Vertex	<input type="radio"/>
FD6, Radius (>0)	1 μm	<input type="radio"/>
As Thin/Surface?	No	



[11, 13, 15, 17, 19, 21] Select **Create/Primitives/Sphere**. → → → ↩ ↩ ↩

[14] Select the same vertex as that in [12]. Click **Generate**. ←

Details View		
Details of Sphere2		
Sphere	Sphere2	
Base Plane	XYPlane	
Operation	Imprint Faces	<input type="radio"/>
Target Bodies	All Bodies	
Origin Definition	Selection	<input type="radio"/>
Origin Selection	Vertex	<input type="radio"/>
FD6, Radius (>0)	2 μm	<input type="radio"/>
As Thin/Surface?	No	

[16] Select the same vertex. Click **Generate**. ←

Details View		
Details of Sphere3		
Sphere	Sphere3	
Base Plane	XYPlane	
Operation	Imprint Faces	<input type="radio"/>
Target Bodies	All Bodies	
Origin Definition	Selection	<input type="radio"/>
Origin Selection	Vertex	<input type="radio"/>
FD6, Radius (>0)	4 μm	<input type="radio"/>
As Thin/Surface?	No	

Details View		
[-] Details of Sphere4		
Sphere	Sphere4	
Base Plane	XYPlane	
Operation	Imprint Faces	<input type="radio"/>
Target Bodies	All Bodies	
Origin Definition	Selection	<input type="radio"/>
Origin Selection	Vertex	<input type="radio"/>
FD6, Radius (>0)	8 μm	<input type="radio"/>
As Thin/Surface?	No	

[18] Select the same vertex. Click **Generate**. ([19] is on the last page.)

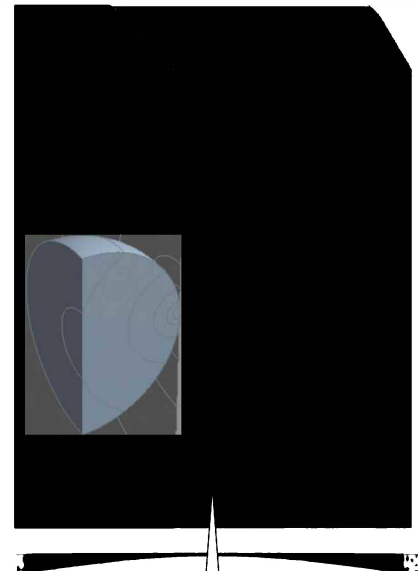
Details View		
[-] Details of Sphere5		
Sphere	Sphere5	
Base Plane	XYPlane	
Operation	Imprint Faces	<input type="radio"/>
Target Bodies	All Bodies	
Origin Definition	Selection	<input type="radio"/>
Origin Selection	Vertex	<input type="radio"/>
FD6, Radius (>0)	16 μm	<input type="radio"/>
As Thin/Surface?	No	

[20] Select the same vertex. Click **Generate**. ([21] is on the last page.)

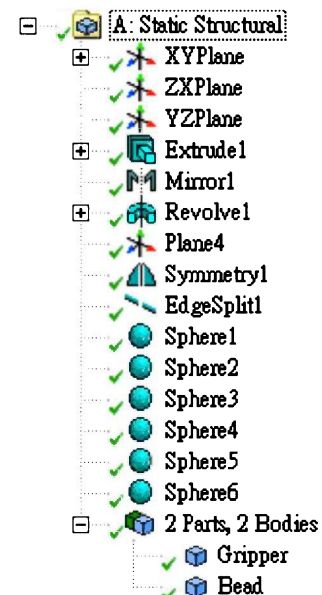
Details View		
[-] Details of Sphere6		
Sphere	Sphere6	
Base Plane	XYPlane	
Operation	Imprint Faces	<input type="radio"/>
Target Bodies	All Bodies	
Origin Definition	Selection	<input type="radio"/>
Origin Selection	Vertex	<input type="radio"/>
FD6, Radius (>0)	32 μm	<input type="radio"/>
As Thin/Surface?	No	

[22] Select the same vertex. Click **Generate**. ↗

[24] The model tree looks like this. Close DesignModeler. #



[23] These spheres **imprint** the faces, dividing the faces into several faces. The spheres don't "slice" the solid bodies; they slice the faces only. We create these faces so we can specify mesh density for each face. ✓



13.3.4 Set Up Material in Engineering Data

Model the Bead as a Rigid Body

[1] The Young's modulus of steel (200 GPa) is 1,000 times that of the polymer material (200 MPa). Whenever two bodies are so different in rigidity, you should model the more rigid one as a rigid body. This not only reduces the problem size (rigid bodies won't deform, they don't need to be meshed), but also eases numerical difficulties during finite element simulations.

A rigid body won't deform; therefore, its Young's modulus and Poisson's ratio are not relevant. For a static simulation, we can assign any material to a rigid body. For dynamic simulation, the mass density should be specified. ↗

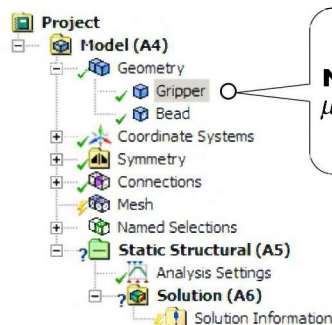
[2] Start up **Engineering Data** and type "Polymer." ↓

Outline of Schematic AZ: Engineering Data					
	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	Material				
3	Polymer				
4	Structural Steel			General Mat	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material				

Properties of Outline Row 4: Polymer					
	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Isotropic Elasticity				
4	Derive from	Young's Modulus and Poisson's Ratio			
5	Young's Modulus	2E+08	Pa		
6	Poisson's Ratio	0.45			
7	Bulk Modulus	1.6667E+09	Pa		
8	Shear Modulus	6.7568E+07	Pa		

[3] Include **Isotropic Elasticity** and input the material properties. Return to **Project Schematic**. #

13.3.5 Set Up Model

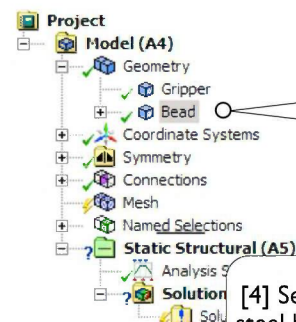


[1] Start up **Mechanical**. Select the μm -kg- μN -s units system. Highlight **Gripper**. ↓

Details of "Gripper"

Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Behavior	None
Material	
Assignment	Polymer
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	
Statistics	

[2] Select **Polymer**. ↗



[3] Highlight Bead. ↓

[4] Select **Rigid**. We model the steel bead as a rigid body to ease computation difficulties (see 13.3.1[1], page 495). ↓

Details of "Bead"

- Graphics Properties**
- Definition**
 - Suppressed: No
 - Stiffness Behavior: Rigid
 - Reference Temperature: By Environment
 - Behavior: None
- Material**
 - Assignment: Structural Steel
- Bounding Box**

computational 13.3.1

[5] In a static simulation, a rigid body doesn't need any material properties. This assignment is simply ignored by Workbench in a static simulation. (Mass density is a required property in dynamic simulations.)

Details of "Frictionless - Gripper To Bead"	
[-] Scope	
Scoping Method	Geometry Selection
Contact	4 Faces
Target	4 Faces
Contact Bodies	Gripper
Target Bodies	Bead
[-] Definition	
Type	Frictionless
Scope Mode	Automatic
Behavior	Program Controlled
Time Contact	Program Controlled
Time Tolerance	2.0087 μm
Suppressed	No
[-] Advanced	
Formulation	Augmented Lagrange
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Stabilization Damping Factor	0.
Pinball Region	Program Controlled
Time Step Controls	None
[-] Geometric Modification	
Interface Treatment	Add Offset, No Ramping
Offset	0. μm
Contact Geometry Correction	None
Target Geometry Correction	None

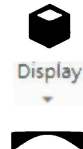
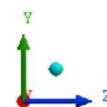
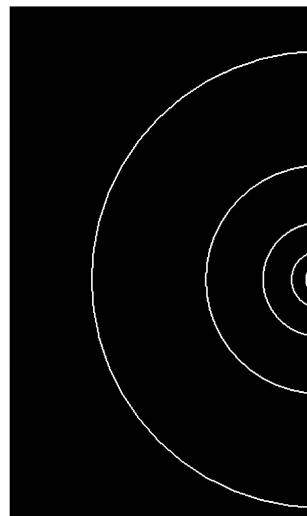
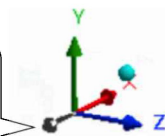
[6] Highlight **Connection/Contact/Contact Region**. Workbench correctly detects and establishes a contact region. You should examine these contact/target faces yourself. Note that Workbench always chooses rigid bodies (if they exist) as **Target Bodies**. ↓

[7] Select **Frictionless**. ↓

[8] Select **Augmented Lagrange** (13.1.10, pages 477-478) for **Formulation**. ✓

[10] Right-click **Bead** in the project tree and select **Hide Body**. Enlarge the contact region like this. Also turn on **Display/Display/Shaded Exterior and Edges** so that the **imprint faces** are visible. ✓

[9] Rotate the view to look at **YZPlane** as shown in [10]. This can be done by rotating the view to approximately look at **YZPlane** and then click the **-X** axis. →



[11, 14, 16, 18, 20, 22] With **Mesh** highlighted, insert a **Sizing**. →

Details of "Face Sizing" - Sizing	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
[-] Definition	
Suppressed	No
Type	Element Size
Element Size	0.125 μm
[-] Advanced	
Default Size	Default
Behavior	Soft

[12] Select the innermost semi-circular face (using **Face Filter**), which has a radius of 1 μm . ↓

[13] Type 0.125 (μm) for **Element Size**. ←

Details of "Face Sizing 2" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face <input type="radio"/>
Definition	
Suppressed	No
Type	Element Size
Element Size	0.25 μm <input type="radio"/>

[15] Select the semi-annular face which has an outer radius of 2 μm . ([16] is on the last page.)

Details of "Face Sizing 3" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face <input type="radio"/>
Definition	
Suppressed	No
Type	Element Size
Element Size	0.5 μm <input type="radio"/>

[17] Select the semi-annular face which has an outer radius of 4 μm . ([18] is on the last page.)

Details of "Face Sizing 4" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face <input type="radio"/>
Definition	
Suppressed	No
Type	Element Size
Element Size	1. μm <input type="radio"/>

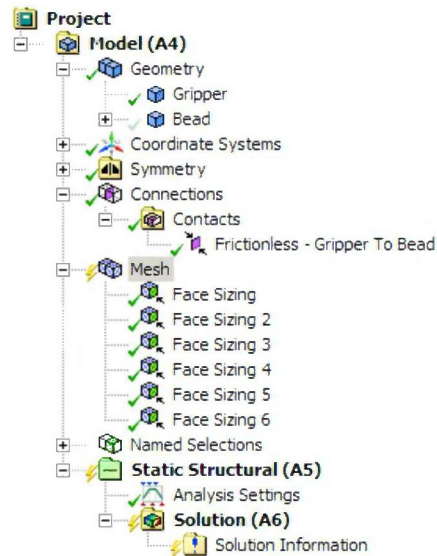
[19] Select the semi-annular face which has an outer radius of 8 μm . ([20] is on the last page.)

Details of "Face Sizing 5" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face <input type="radio"/>
Definition	
Suppressed	No
Type	Element Size
Element Size	2. μm <input type="radio"/>

[21] Select the semi-annular face which has an outer radius of 16 μm . ([22] is on the last page.)

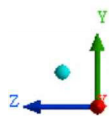
Details of "Face Sizing 6" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face <input type="radio"/>
Definition	
Suppressed	No
Type	Element Size
Element Size	4. μm <input type="radio"/>

[23] Select the semi-annular face which has an outer radius of 32 μm .



[24] Highlight **Mesh** and select **Mesh/Preview/Surface Mesh** for a quick preview of the surface mesh. ↩





[25] Rotate the view to look at **YZPlane**. →

Details of "Face Sizing 7" - Sizing		
☏		
Scope		
Scoping Method	Geometry Selection	
Geometry	1 Face	○
Definition		
Suppressed	No	
Type	Element Size	
Element Size	0.125 μm	○

[28] Select the innermost semi-circular face, which has a radius of 1 μm . →

Details of "Face Sizing 8" - Sizing		
☏		
Scope		
Scoping Method	Geometry Selection	
Geometry	1 Face	○
Definition		
Suppressed	No	
Type	Element Size	
Element Size	0.25 μm	○

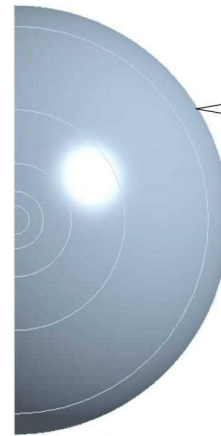
[30] Select the annular face which has an outer radius of 2 μm . →

Details of "Face Sizing 9" - Sizing		
☏		
Scope		
Scoping Method	Geometry Selection	
Geometry	1 Face	○
Definition		
Suppressed	No	
Type	Element Size	
Element Size	0.5 μm	○

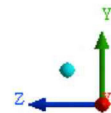
[32] Select the annular face which has an outer radius of 4 μm . →

Details of "Face Sizing 10" - Sizing		
☏		
Scope		
Scoping Method	Geometry Selection	
Geometry	1 Face	○
Definition		
Suppressed	No	
Type	Element Size	
Element Size	1. μm	○

[34] Select the annular face which has an outer radius of 8 μm . →



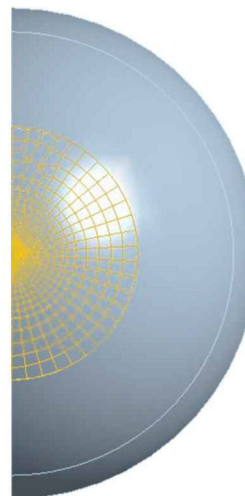
[26] Hide **Gripper** and show **Bead**. Enlarge **Bead** like this. ↓

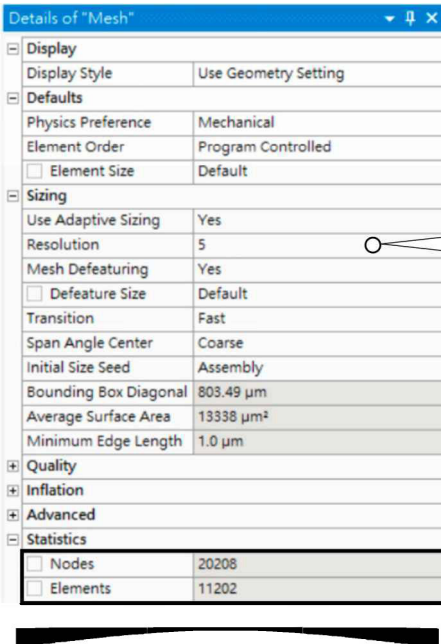


[27, 29, 31, 33] With **Mesh** highlighted, insert a **Sizing**.
← ← ← ←



[35] Highlight **Mesh** and select **Mesh/Preview/Surface Mesh** for a quick preview. ↵

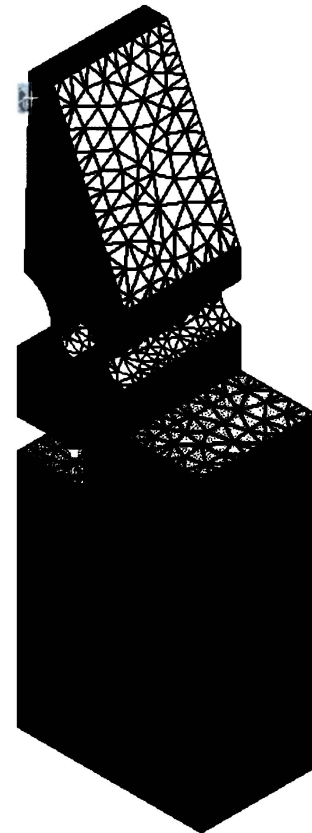
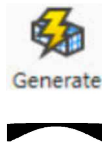




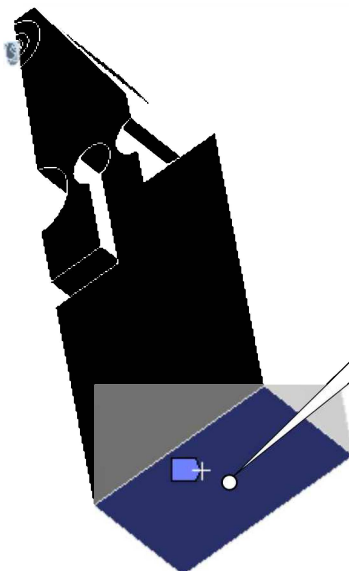
[36] Select **5** for **Resolution**. →

[38] Mesh count. #

[37] Show **Gripper**.
Generate mesh. ✓



13.3.6 Set Up Environment Conditions



[1] Set up a **Fixed Support** on this face. ↩

Details of "Fixed Support"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Fixed Support
Suppressed	No

[2] Select Support/**Remote Displacement**. ↓

[3] Rotate the view to look at **XYPlane** and select a face (any face) of the bead. ↓

[4] And click **Apply**. ↓

[5] Workbench calculates the geometric center of the face. In this case, these numbers are not relevant. For a rigid body, fixing any point means fixing the entire rigid body. ↓

Details of "Remote Displacement"

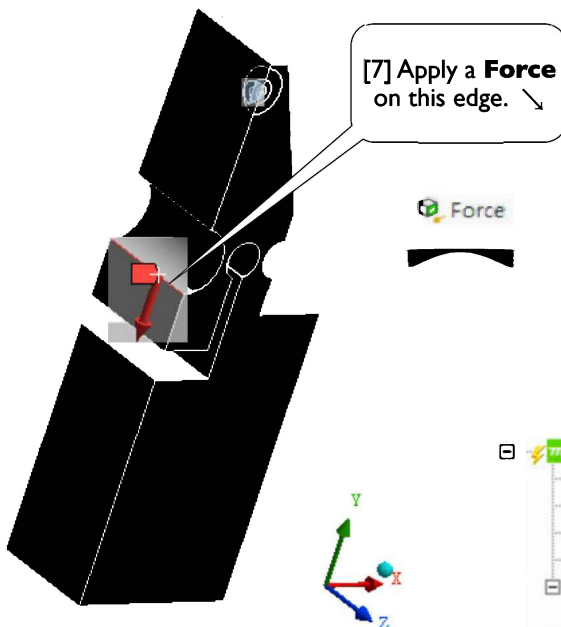
Scope		
Scoping Method	Geometry Selection	
Geometry	1 Face	
Coordinate System	Global Coordinate System	
<input type="checkbox"/> X Coordinate	5.3861 μm	
<input type="checkbox"/> Y Coordinate	713.67 μm	
<input type="checkbox"/> Z Coordinate	7.5988e-017 μm	
Location	Click to Change	
Definition		
Type	Remote Displacement	
<input type="checkbox"/> X Component	0. μm (ramped)	<input type="radio"/>
<input type="checkbox"/> Y Component	0. μm (ramped)	<input type="radio"/>
<input type="checkbox"/> Z Component	0. μm (ramped)	<input type="radio"/>
<input type="checkbox"/> Rotation X	0. ° (ramped)	<input type="radio"/>
<input type="checkbox"/> Rotation Y	0. ° (ramped)	<input type="radio"/>
<input type="checkbox"/> Rotation Z	0. ° (ramped)	<input type="radio"/>
Suppressed	No	
Behavior	Rigid <input type="radio"/>	
Advanced		

Remote Displacement

[6] A rigid body is internally represented as a node of 6 degrees of freedom, three translational and three rotational. The node is located at the geometric center of the rigid body. All environment conditions are applied at the geometric center of the rigid body. Using **Remote Displacement**, we can let Workbench transform the environment conditions to the node.

When we select a face [3-4] and apply a **Remote Displacement**, Workbench calculates the geometry center for that face, and the specified displacement is applied at that location. That displacement is then transformed to the geometry center of the rigid body.

In our case, we arbitrarily select a face [3-4]. The geometric center of that face will have zero displacements and zero rotations. These conditions will be transformed to the geometry center of the rigid body. The result is that the entire rigid body is fixed, both translationally and rotationally. That's what we intend for this rigid body. ↵



[7] Apply a **Force** on this edge. ↘

Force

Static Structural (A5)

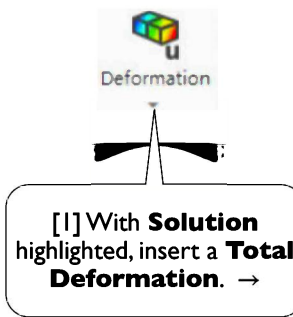
- Analysis Settings
- Fixed Support
- Remote Displacement
- Force
- Solution (A6)
- Solution Information

Details of "Force"

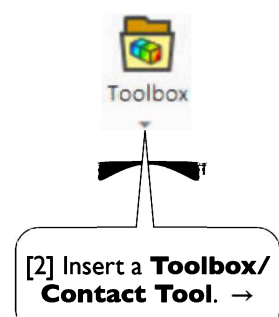
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Type	Force
Define By	Components
Applied By	Surface Effect
Coordinate System	Global Coordinate System
<input type="checkbox"/> X Component	0. μN (ramped)
<input checked="" type="checkbox"/> Y Component	-10. μN (ramped)
<input type="checkbox"/> Z Component	0. μN (ramped)
Suppressed	No

[8] The magnitude is 10 μN downward. Since the simulation model is only a quarter of the real model, only a quarter of the total load (40 μN) is applied. #

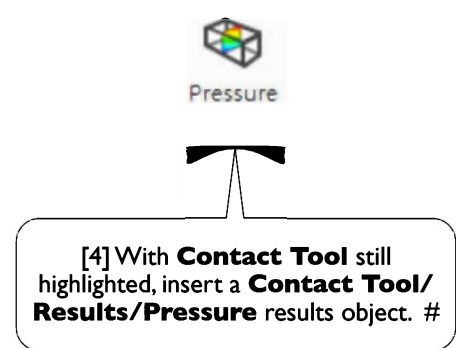
13.3.7 Set Up Result Objects



[1] With **Solution** highlighted, insert a **Total Deformation**. →



[2] Insert a **Toolbox/Contact Tool**. →



[4] With **Contact Tool** still highlighted, insert a **Contact Tool/Results/Pressure** results object. #

[3] Select **Worksheet** (default) for **Scoping Method** and leave the settings in **Worksheet** (to the right side of the GUI, as shown below) unchanged. ✓

Details of "Contact Tool"

Scope	
Scoping Method	Worksheet

Contact Tool

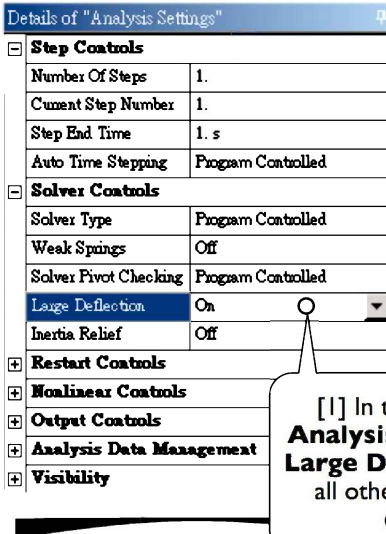
Contacts Selection: All Contacts [Add] [Remove]

Contact Side: Both [Apply]

For additional options, please visit the context menu for this table (right mouse button)

Name	Contact Side
<input checked="" type="checkbox"/> Frictionless - Gripper To Bead	Both

13.3.8 Solve the Model and View the Results

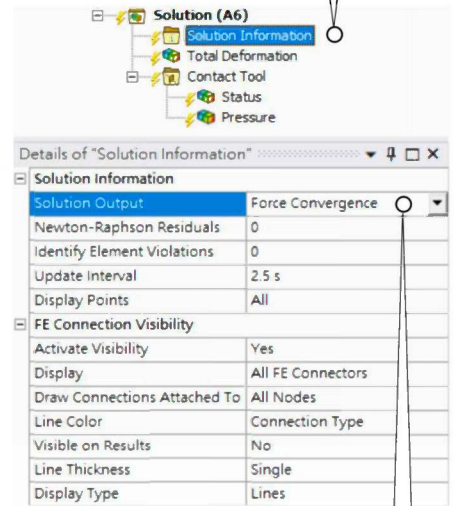


[1] In the details view of **Analysis Settings**, turn on **Large Deflection** and leave all other settings at their defaults. ↗

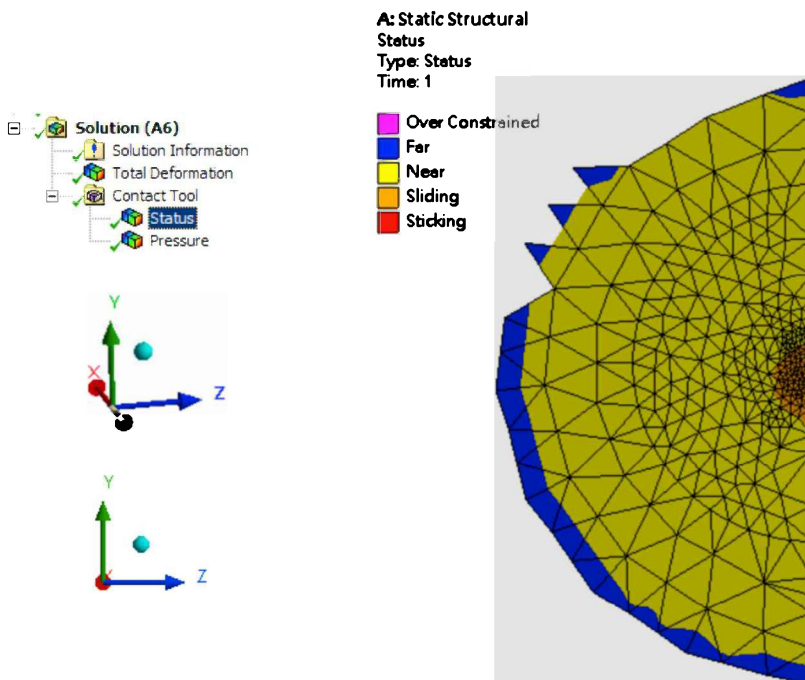


[4] Click **Solve**. ↘

[2] Highlight **Solution Information**. ↓



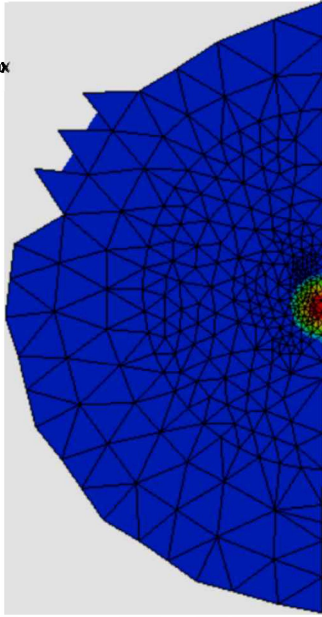
[3] Select **Force Convergence**. ←



[5] Hide **Bead**. Highlight **Contact Tool/Status**. View from the **-X** side. Enlarge the contact region to view the contact status. ↶

A: Static Structural
Pressure
Type: Pressure
Unit: MPa
Time: 1

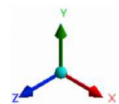
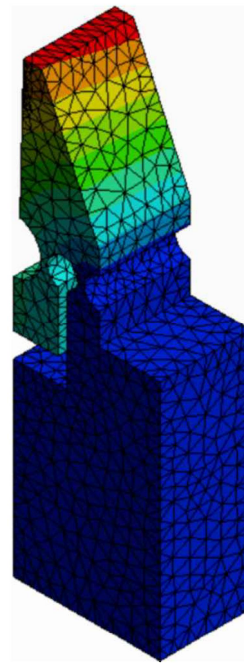
4.7196 Max
4.1952
3.6708
3.1464
2.622
2.0976
1.5732
1.0488
0.5244
0 Min



[6] Highlight **Contact Tool/Pressure** to view the contact pressure. ↓

A: Static Structural
Total Deformation
Type: Total Deformation
Unit: μm
Time: 1

0.03761 Max
0.033431
0.029252
0.025073
0.020894
0.016715
0.012537
0.0083577
0.0041789
0 Min



Wrap Up

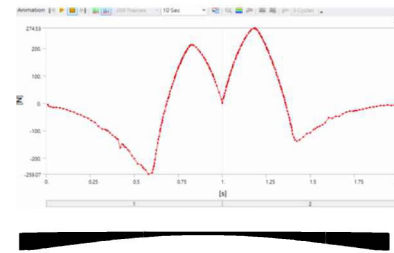
[7] Animate **Total Deformation** (using **Auto Scale**).
Save the project and exit Workbench. #

References

1. Chang, R. J., Lin, Y. C., Shiu, C. C., and Hsieh, Y. T., "Development of SMA-Actuated Microgripper in Micro Assembly Applications," IECON, IEEE, Taiwan, 2007.
2. Shih, P. W., *Applications of SMA on Driving Micro-gripper*, MS Thesis, NCKU, ME, Taiwan, 2005.

Section 13.4

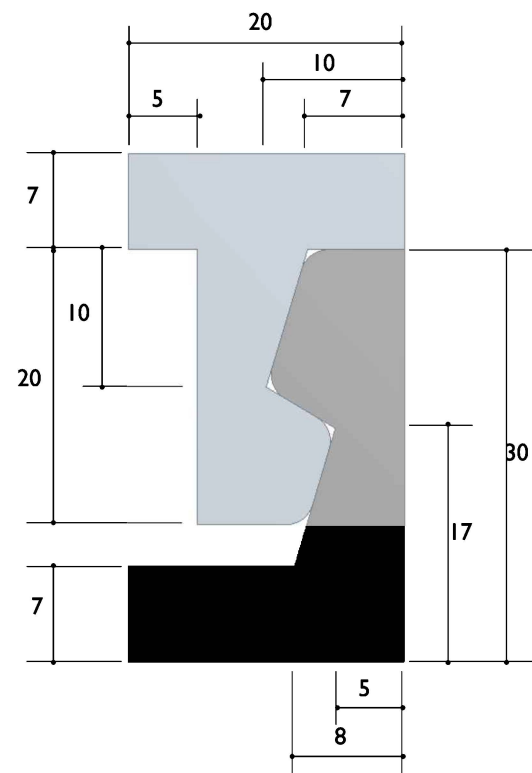
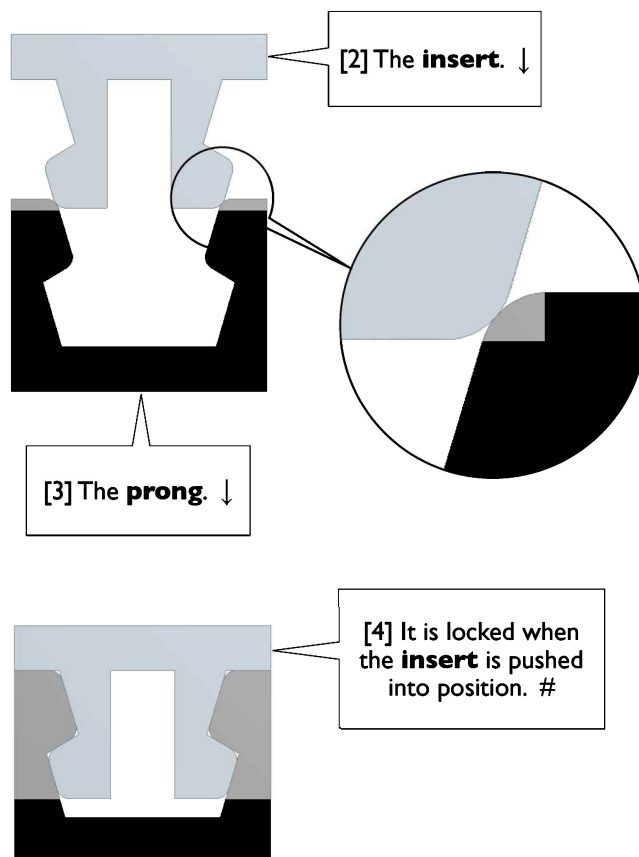
Snap Lock



13.4.1 About the Snap Lock

[1] The snap lock consists of two parts: the **insert** [2] and the **prong** [3]. It is locked when the **insert** is pushed into position [4]. The snap lock has a thickness of 5 mm and is made of a plastic material with a Young's modulus of 2.8 GPa and a Poisson's ratio of 0.35. The coefficient of friction between the parts is 0.1. The purpose of this simulation is to find out the force required to push the **insert** into the position and the force required to pull it out.

We will model the problem as a plane stress problem. Due to the symmetry, only one half of the snap lock is modeled for the simulation. ↓



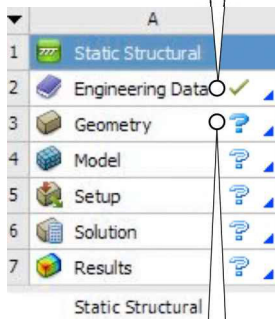
Unit: mm.

All fillets have a radius of 2 mm.

13.4.2 Start Up

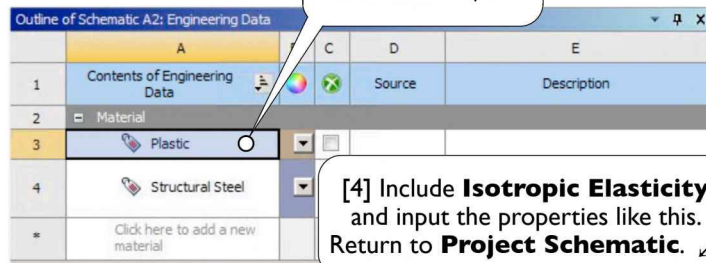
[1] Launch Workbench. Create a **Static Structural** analysis system. Save the project as **Snap**. ↓

[2] Double-click **Engineering Data**. →



[5] Start up DesignModeler. #

[3] Add a material **Plastic**. ↓



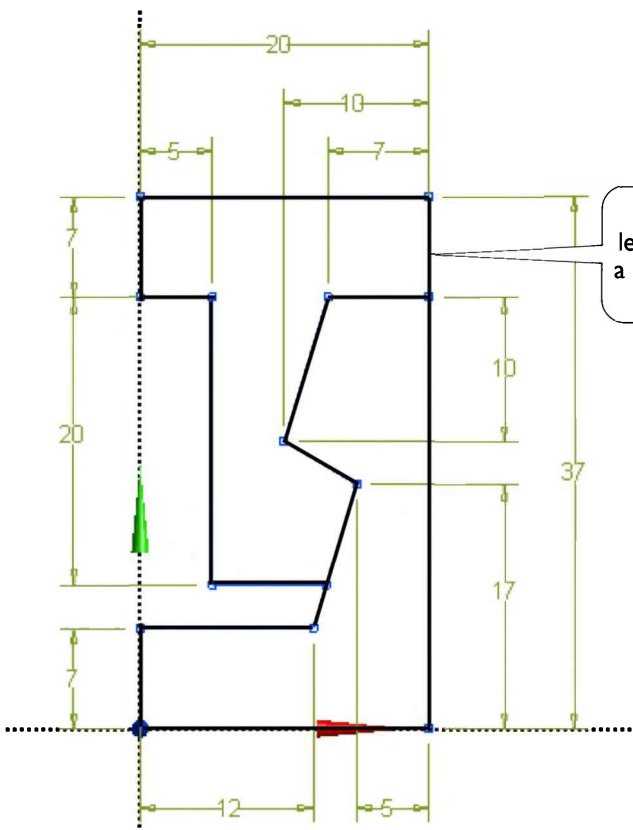
[4] Include **Isotropic Elasticity** and input the properties like this. Return to **Project Schematic**. ✓

Properties of Outline Row 4: Plastic				
	A	B	C	D E
1	Property	Value	Unit	
2	Material Field Variables	Table		
3	Isotropic Elasticity			
4	Derive from	Young's Modulus and Poisson's Ratio		
5	Young's Modulus	2.8E+09	Pa	
6	Poisson's Ratio	0.35		
7	Bulk Modulus	3.111E+09	Pa	
8	Shear Modulus	1.037E+09	Pa	

13.4.3 Create Geometry in DesignModeler

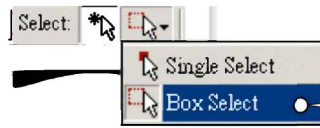
[1] Select **Millimeter** as the length unit. On **XYPlane**, create a sketch like this. Make sure all the lines are blue-colored. ↘

[2] Click **New Sketch** to create a new sketch (**Sketch2**) on the same plane. ↵

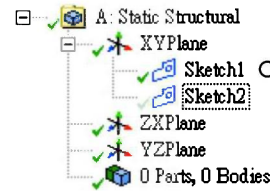




[3] Duplicate all the existing entities to **Sketch2**. (See next step.) ↗

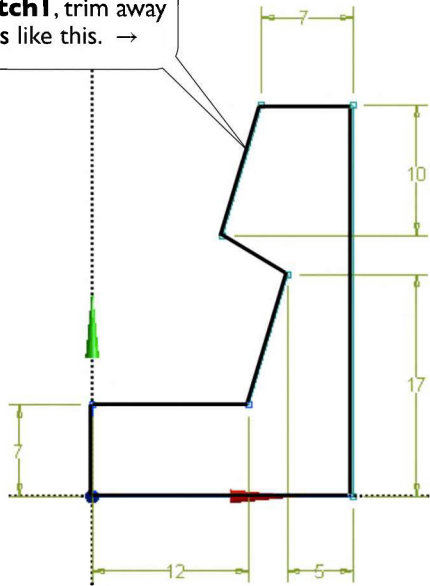


[4] You may use **Box Select** to select all the entities. After selecting all entities, right-click-select **Duplicate Selection**. Now, **Sketch1** and **Sketch2** are identical. ↓

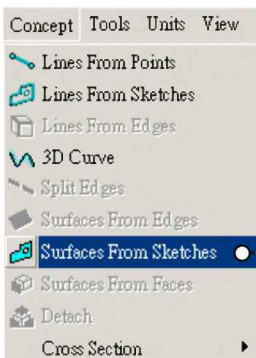
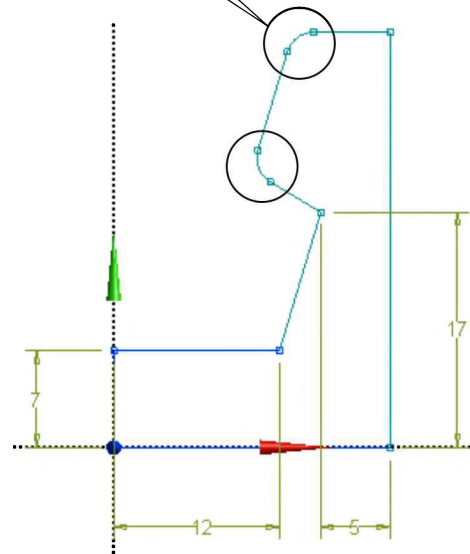


[5] Hide **Sketch2** and Select **Sketch1**. ←

[6] In **Sketch1**, trim away segments like this. →



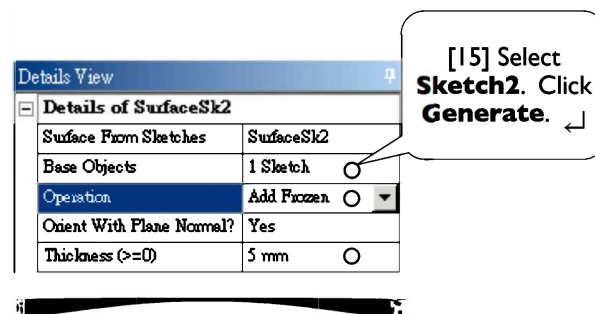
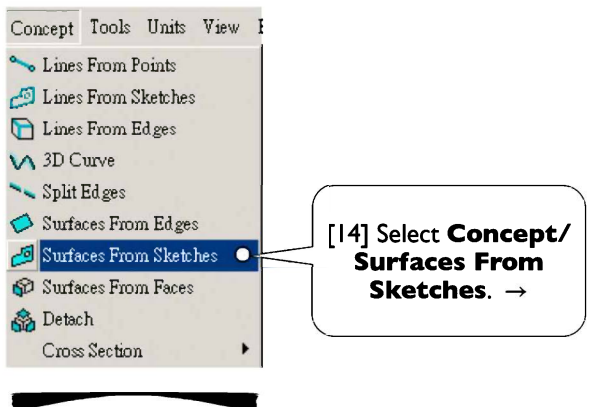
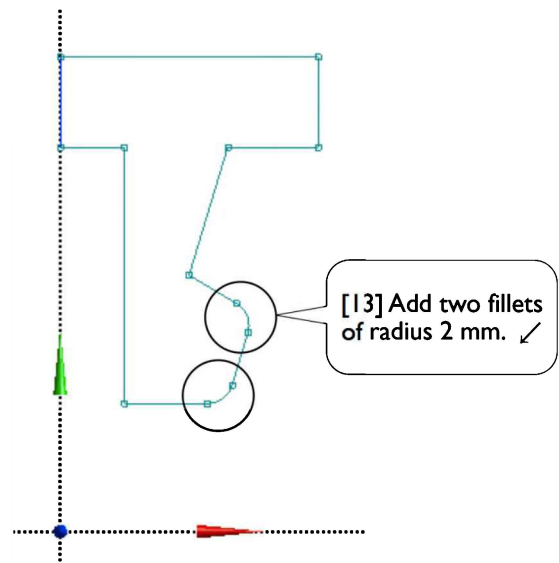
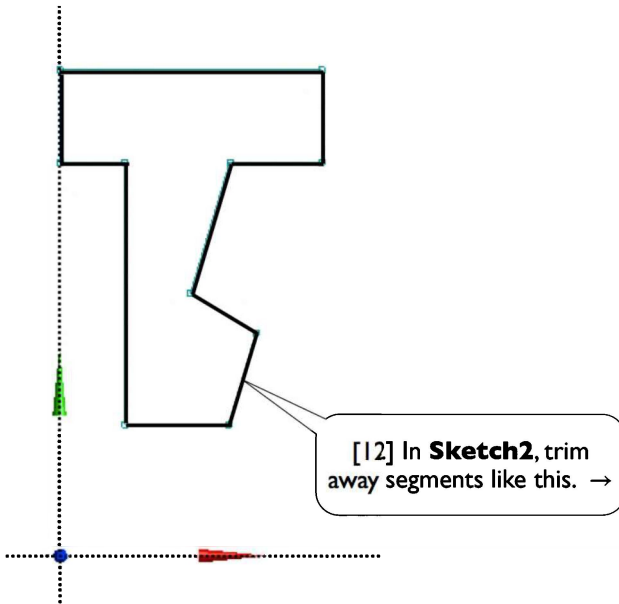
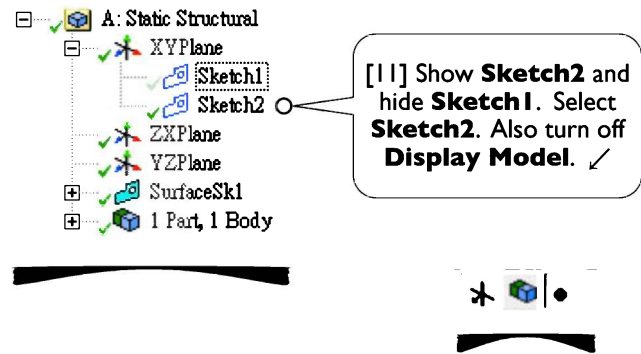
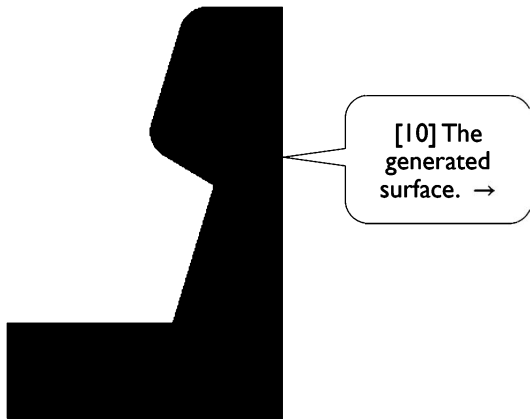
[7] Add two fillets of radius 2 mm. ✓

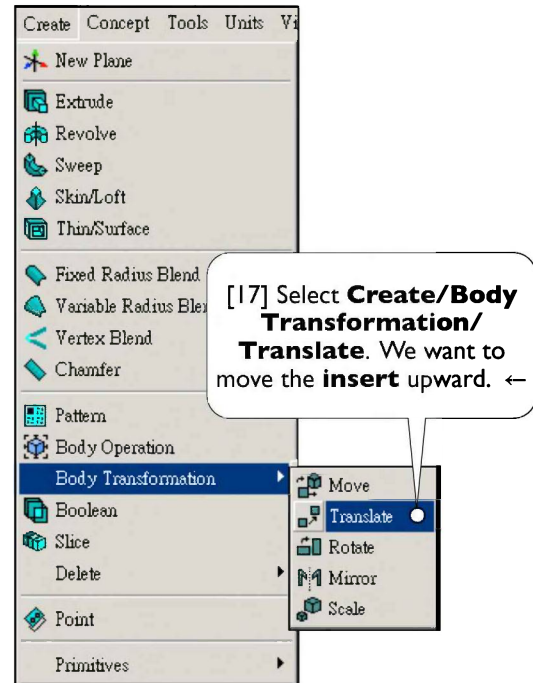
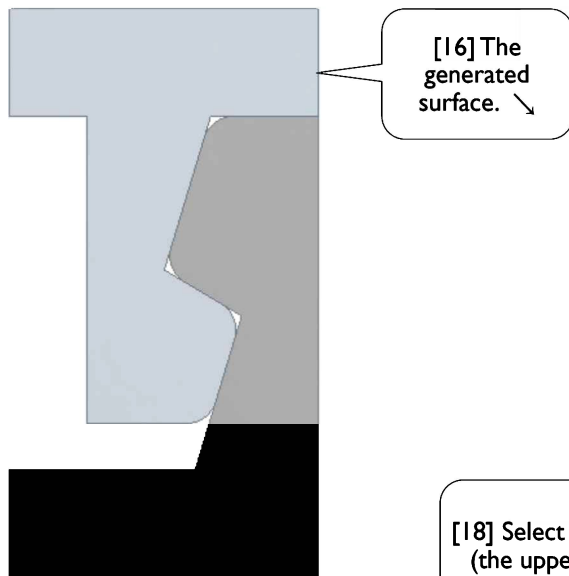


[8] Select **Concept/ Surfaces From Sketches**. →

Details View	
Details of SurfaceSk1	
Surface From Sketches	SurfaceSk1
Base Objects	1 Sketch
Operation	Add Material
Orient With Plane Normal?	Yes
Thickness (≥ 0)	5 mm

[9] Select **Sketch1**. Click **Generate**. ↶





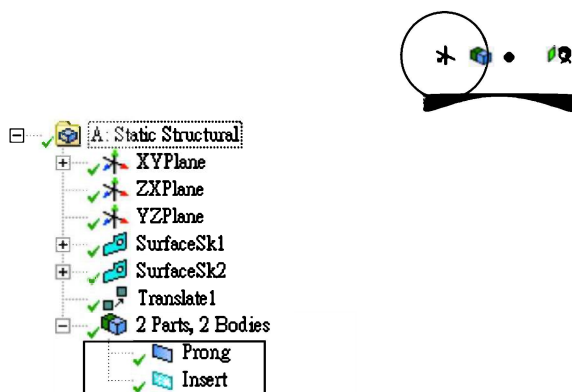
Details View

Details of Translate1	
Translate	Translate1
Preserve Bodies?	No <input type="radio"/>
Bodies	1 <input type="radio"/>
Direction Definition	Coordinates <input type="radio"/>
FD3, X Offset	0 mm
<input checked="" type="checkbox"/> FD4, Y Offset	18.56 mm <input type="radio"/>
FD5, Z Offset	0 mm

[18] Select the **insert** (the upper part). ↓

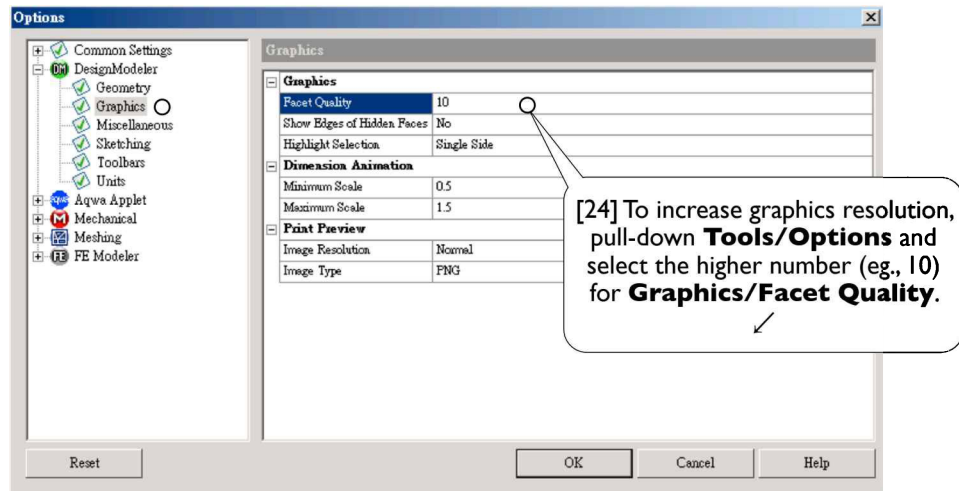
[19] Offset the **insert** upward so that it merely touches the **prongs**. By several trials, you should come up with a number like this. Click **Generate**. Turn off **Display Plane**. ↗

[21] **Insert**. ↓



[23] Enlarge to make sure they merely touch each other. If they don't, change the value in [19]. To make this task more precise, you may select a higher **Facet Quality** (see next step). ↶

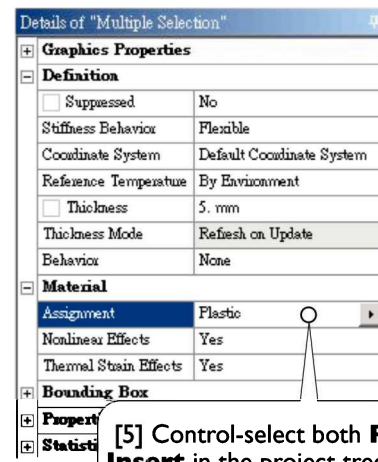
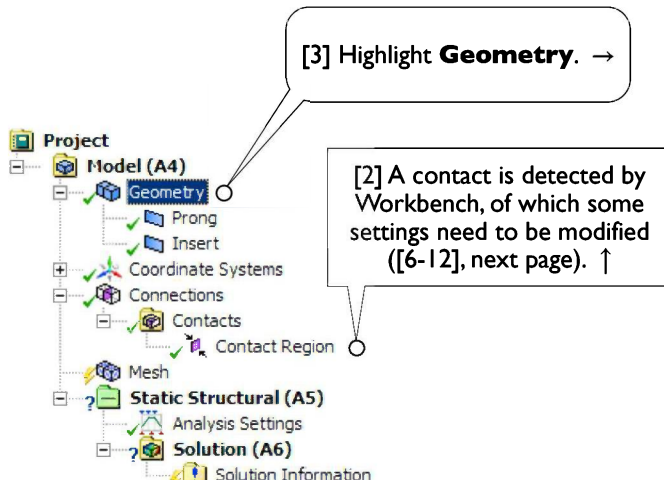
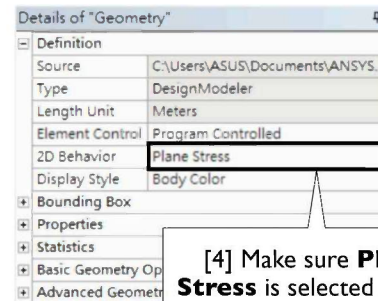
[22] **Prong**. ↑

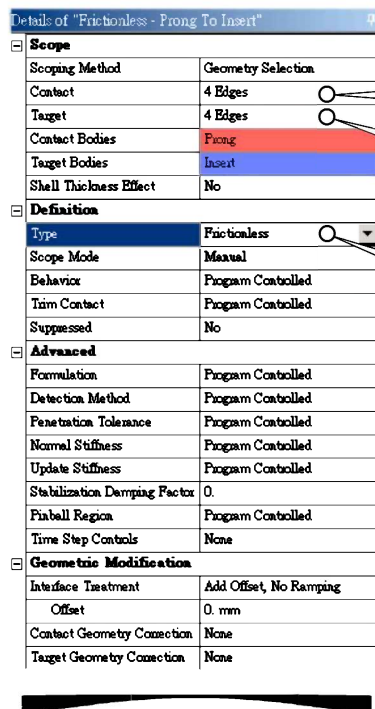


[25] Close DesignModeler. Before attaching the geometry to **Mechanical**, select **2D** option (3.1.4[4-6], page 112). #

13.4.4 Set Up Model

[1] Start up **Mechanical**. Select the unit system **mm-kg-N-s**. ↓

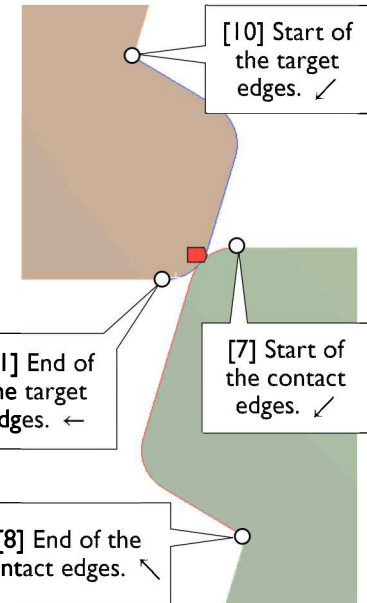
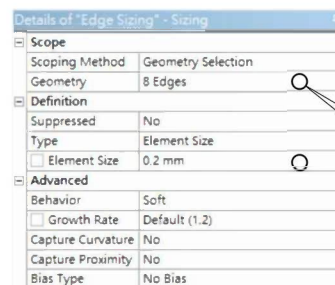




[6] Highlight **Contact Region** (see [2], last page) and reselect the 4 edges shown in [7-8] as **Contact**. ↘

[9] Reselect the 4 edges shown in [10-11] as **Target**. ↗

[12] Select **Frictionless**. (We neglect friction for the first run.) Leave the other settings at their default values. ✓



[11] End of the target edges. ←

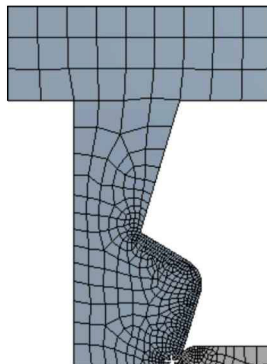
[7] Start of the contact edges. ✓

[8] End of the contact edges. ↘

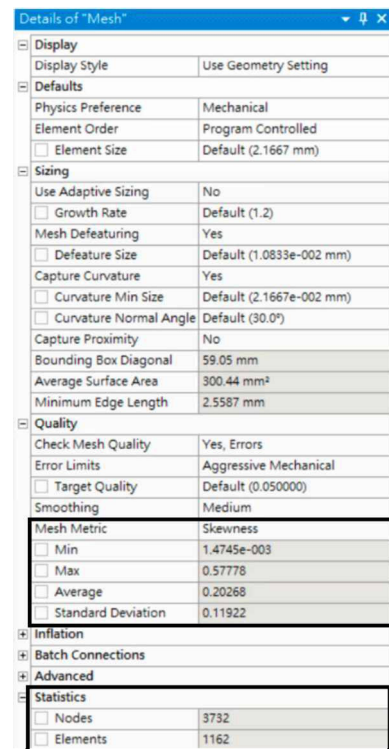
[14] Control-select all edges in **Contact Region** [7-8, 10-11], 8 edges in total. ✓



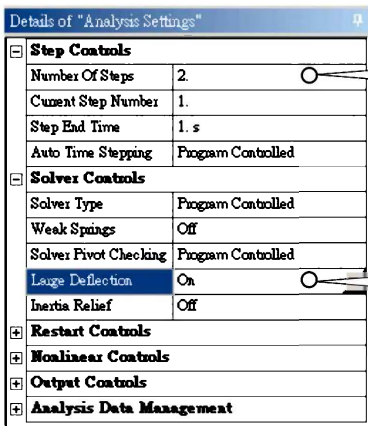
[13] With **Mesh** highlighted, select **Mesh/Sizing**. ↗



[15] Generate mesh. #



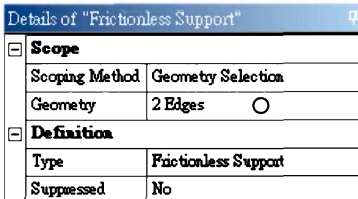
13.4.5 Set Up Analysis Settings



[1] In **Analysis Settings**, type 2 for **Number of Steps**. We will apply two steps of displacement on the insert: downward and then upward (see 13.4.6[4-6], this and next pages). ↓

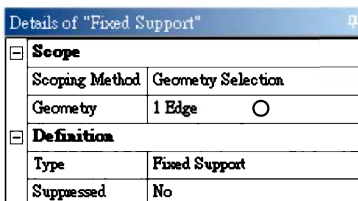
[2] Turn on **Large Deflection**. Leave other settings their default values. By default, step 2 has the same settings as step 1. #

13.4.6 Set Up Environment Conditions



[2] Set up a **Frictionless Support** on this edge and the edge in the next step. ↓

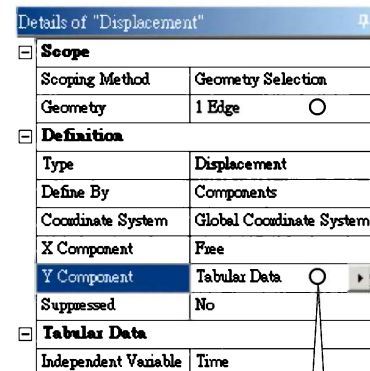
[3] And this edge. For solid elements, a **Frictionless Support** is equivalent to a symmetry condition. ↗



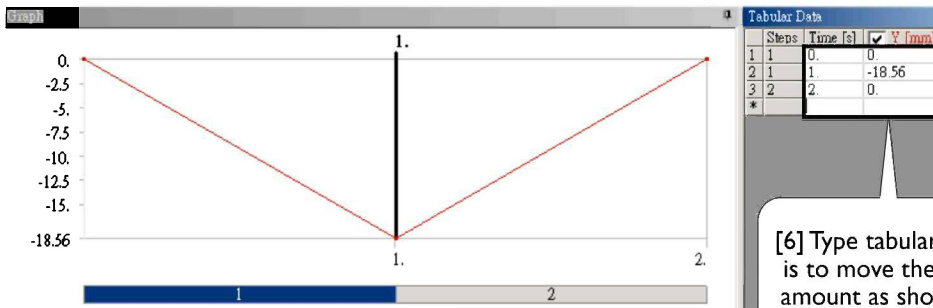
[1] Set up a **Fixed Support** on this edge. ↑



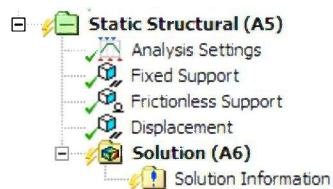
[4] Set up a **Displacement** on this edge (see [5-6]). ↓



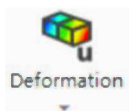
[5] Select **Tabular** for **Y Component**. ↵



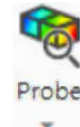
[6] Type tabular data like this. The first step is to move the **insert** downward with an amount as shown in 13.4.3[19] (page 512). The second step is to move the **insert** upward back to the original position. #



13.4.7 Set Up Result Objects



[1] With **Solution** highlighted, insert a **Total Deformation**. →



[2] Insert a **Probe/Force Reaction**. ✓

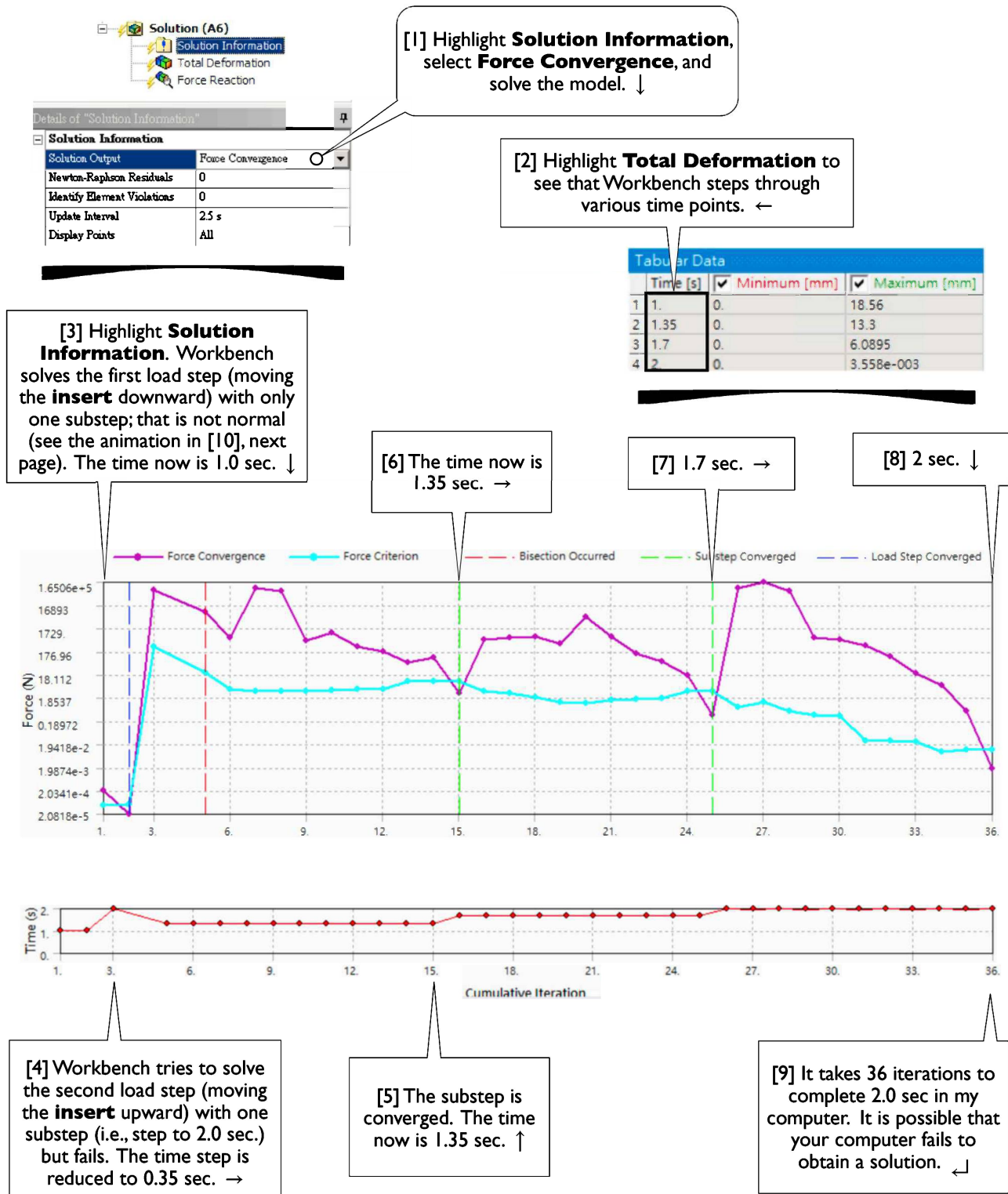
Details of "Force Reaction"

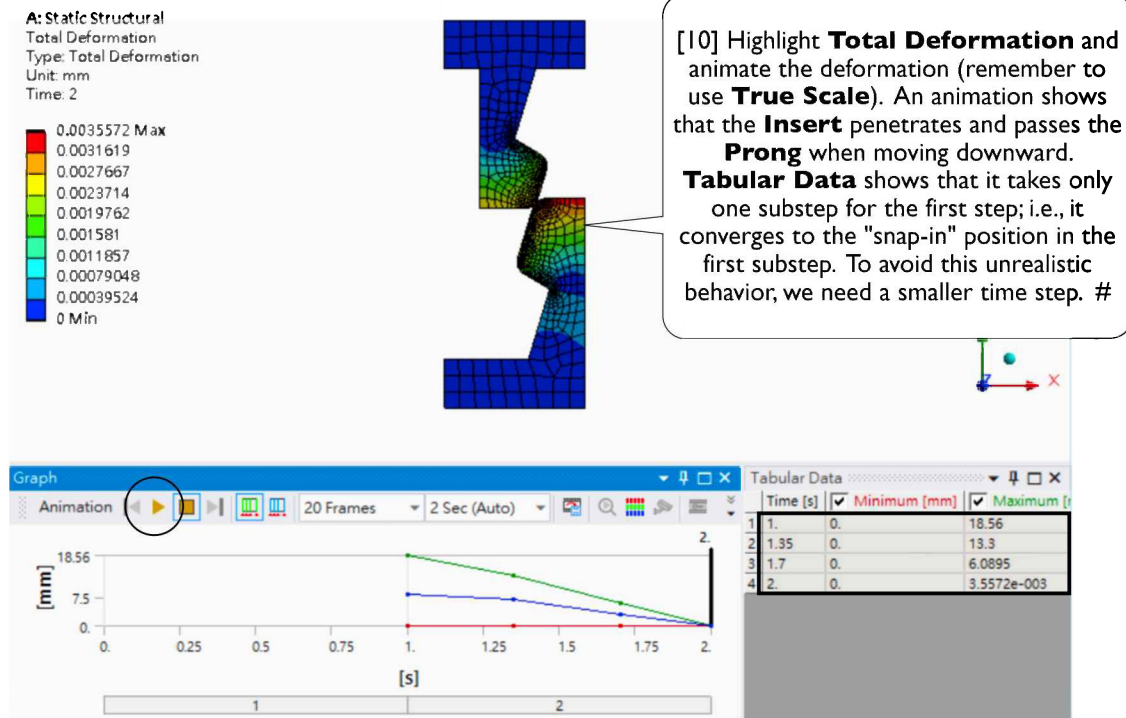
Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Displacement <input type="radio"/>
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	Y Axis <input type="radio"/>
Display Time	End Time
Results	
Maximum Value Over Time	
Y Axis	
Minimum Value Over Time	
Y Axis	
Information	

[3] Select **Displacement**. Workbench will calculate the forces required for the displacement specified in 13.4.6[6], this page. #



13.4.8 Solve the Model and View the Results





13.4.9 Reduce Time Steps and Solve Again

[1] Set up **Analysis Settings** like this for the first step. →

[2] Set up **Analysis Settings** like this for the second step (you may click the second step in **Graph** or **Tabular Data**). ↓

[3] Turn on **Carry Over Time Step** to use the time step value of the last substep as the initial time step value for this step. ↵

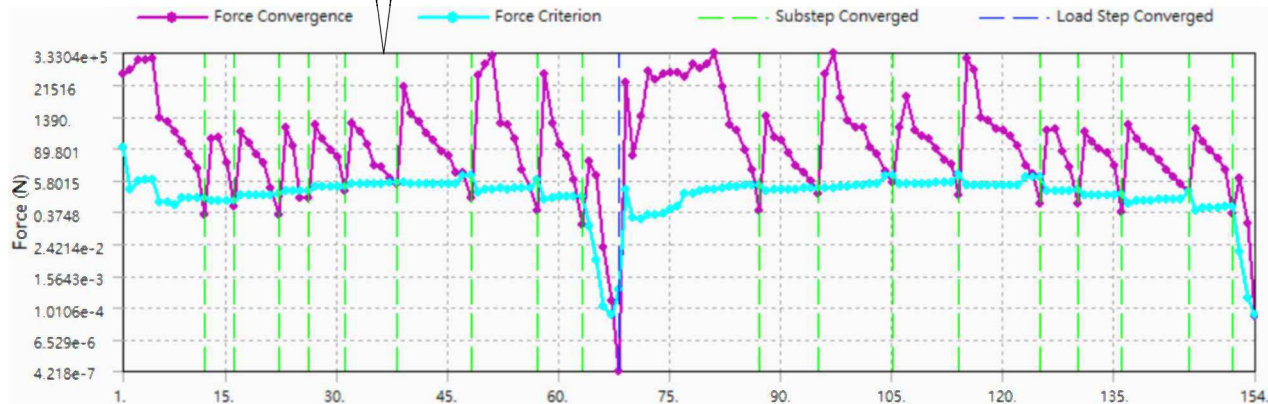
Details of "Analysis Settings"

Step Controls	
Number Of Steps	2.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	On
Define By	Time
Initial Time Step	0.1 s
Minimum Time Step	1.e-002 s
Maximum Time Step	0.1 s
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Restart Controls	
Nonlinear Controls	
Output Controls	
Analysis Data Management	
Visibility	

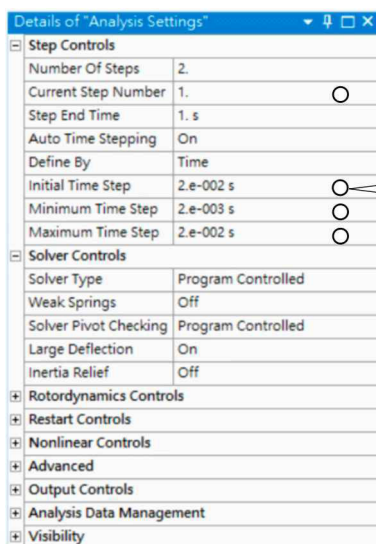
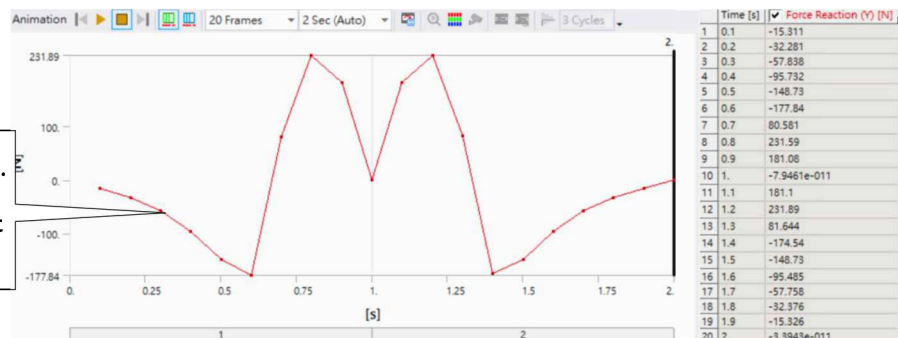
Details of "Analysis Settings"

Step Controls	
Number Of Steps	2.
Current Step Number	2.
Step End Time	2. s
Auto Time Stepping	On
Define By	Time
Carry Over Time Step	On
Minimum Time Step	1.e-002 s
Maximum Time Step	0.1 s
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Restart Controls	
Nonlinear Controls	
Output Controls	
Analysis Data Management	
Visibility	

[4] Highlight **Solution Information** and solve again. My computer successfully solves the model. If your computer doesn't, don't mind. ↓

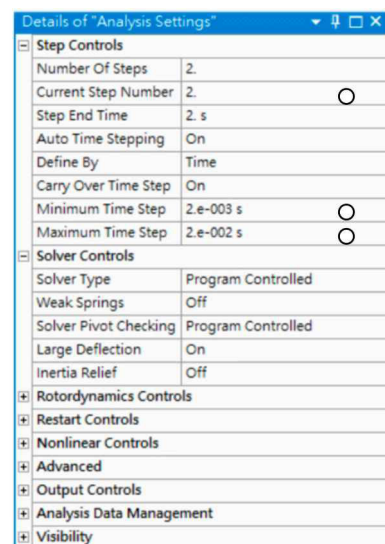


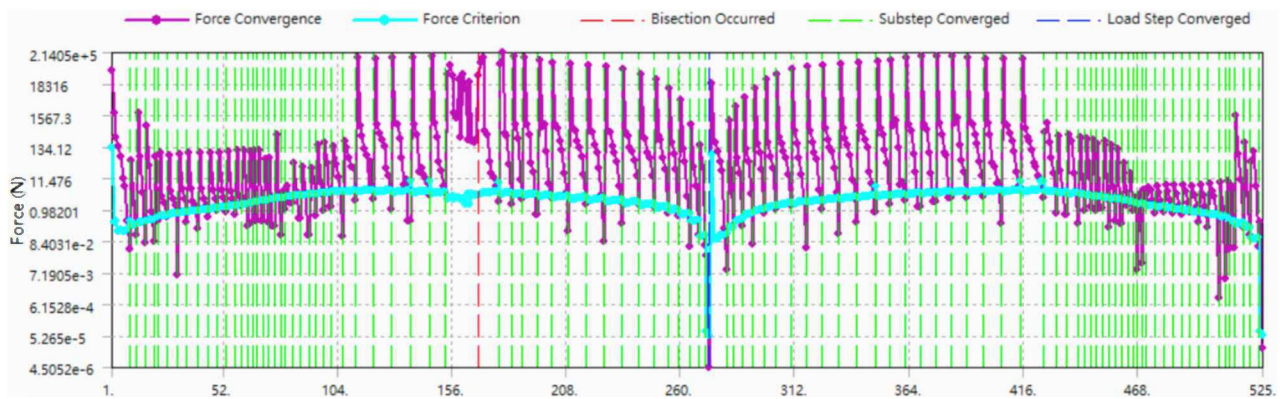
[5] Highlight **Force Reaction**. Even my computer solves the model, the solution curve is not smooth. ↓



[6] Reduce **Time Step**. ↓

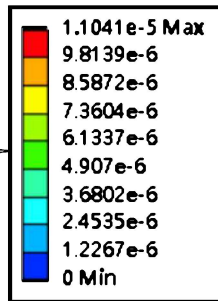
[7] Repeat this for the second step. ↩



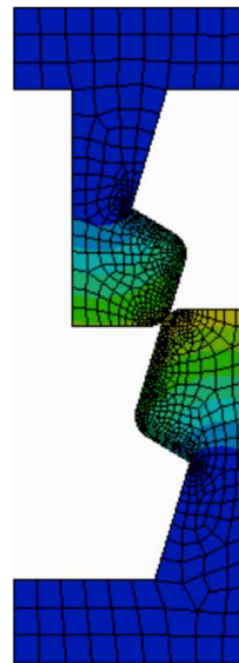


[8] Highlight **Solution Information** and solve the model. It takes many iterations to complete the simulation. ✓

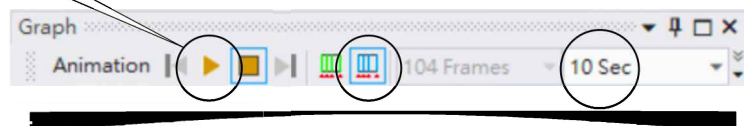
A: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 2

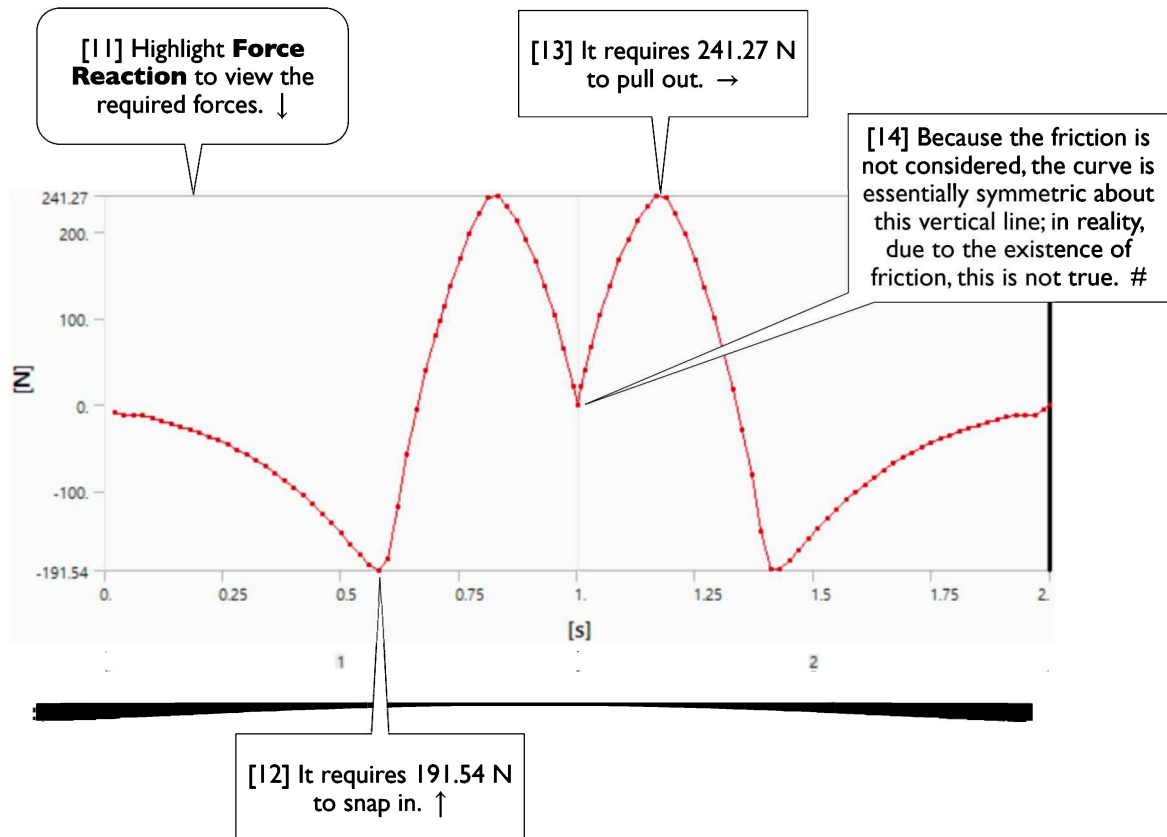


[9] Highlight **Total Deformation**. At the end time (2 sec), the displacements essentially return to zeros. ↓

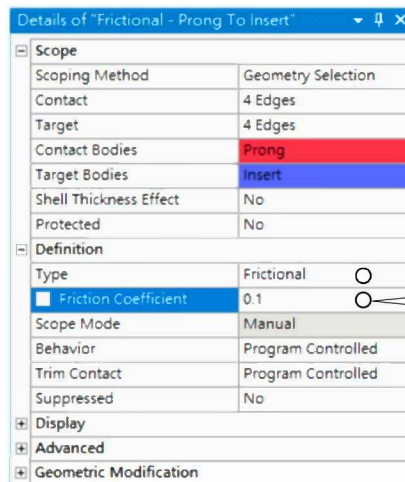
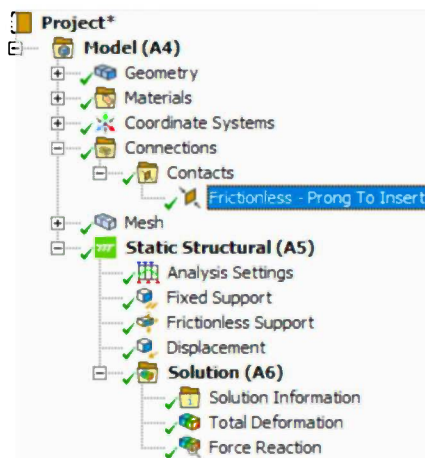


[10] Animate the displacements. ↶



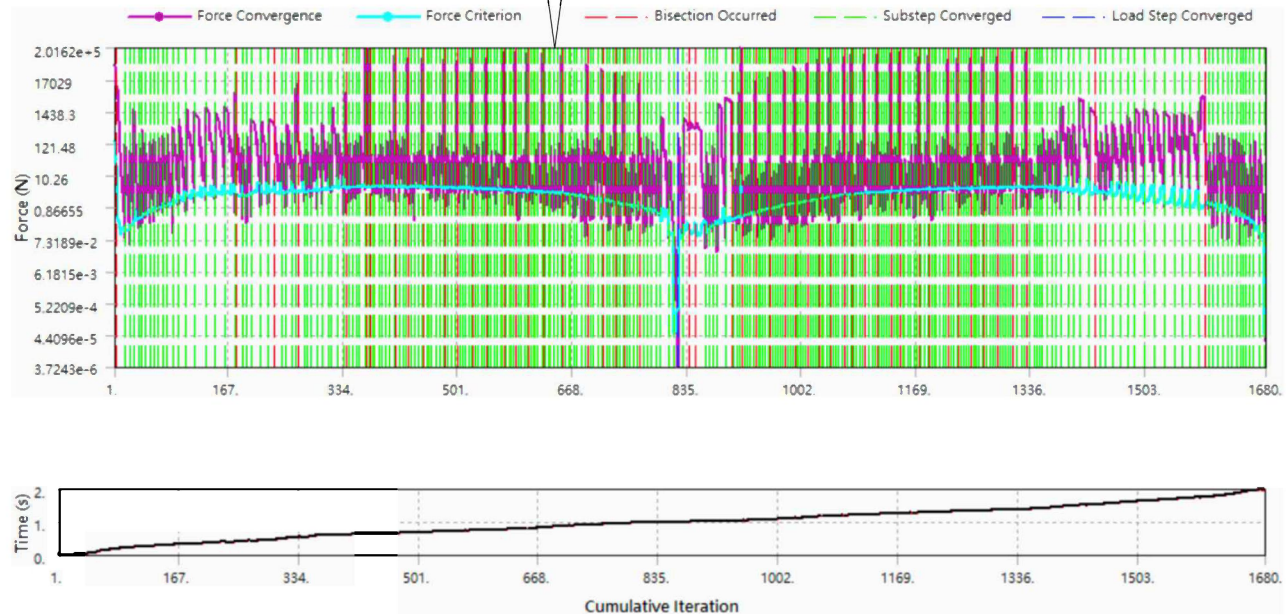


13.4.10 Simulation with Frictional Model



[1] Highlight **Frictionless - Prong To Insert**, change the **Type** to **Frictional**, and type 0.1 for **Friction Coefficient**. ↵

[2] Highlight **Solution Information** and solve again. It takes even more iterations to complete the simulation. ✓



[3] Select **Solver Output** to view the text information. ↓

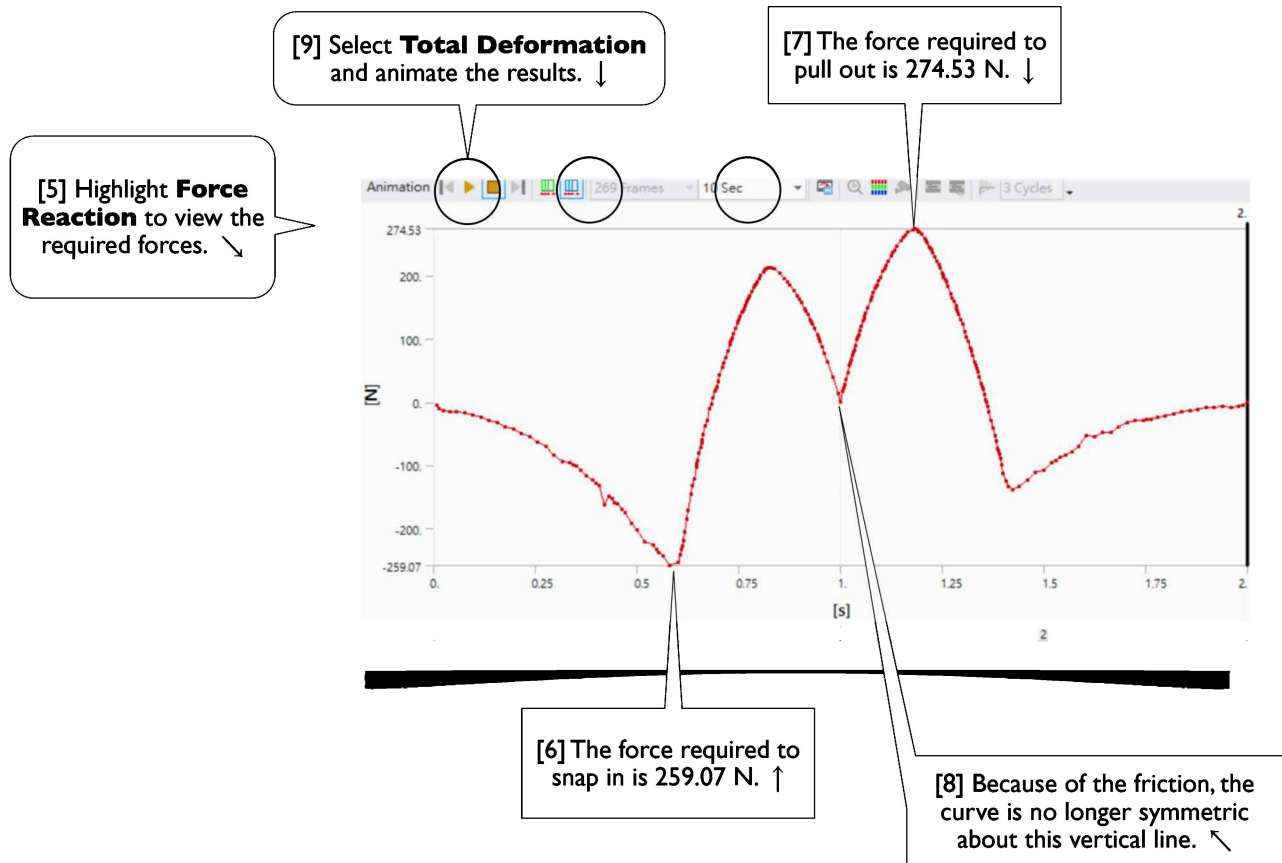
Details of "Solution Information"	
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Identify Element Violations	0
Update Interval	25 s
Display Points	All

```

EQUIL ITER 3 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -0.1686E-01
DISP CONVERGENCE VALUE = 0.1397E-01 CRITERION= 0.3957E-01 <<< CONVERGED
LINE SEARCH PARAMETER = 0.8285 SCALED MAX DOF INC = -0.1397E-01
FORCE CONVERGENCE VALUE = 26.64 CRITERION= 0.8279E-01
EQUIL ITER 4 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.7403E-02
DISP CONVERGENCE VALUE = 0.7403E-02 CRITERION= 0.4038E-01 <<< CONVERGED
LINE SEARCH PARAMETER = 1.000 SCALED MAX DOF INC = 0.7403E-02
FORCE CONVERGENCE VALUE = 4.685 CRITERION= 0.9630E-01
EQUIL ITER 5 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.4607E-02
DISP CONVERGENCE VALUE = 0.4254E-02 CRITERION= 0.4120E-01 <<< CONVERGED
LINE SEARCH PARAMETER = 0.9234 SCALED MAX DOF INC = 0.4254E-02
FORCE CONVERGENCE VALUE = 2.902 CRITERION= 0.1038
EQUIL ITER 6 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -0.2176E-03
DISP CONVERGENCE VALUE = 0.2176E-03 CRITERION= 0.4204E-01 <<< CONVERGED
LINE SEARCH PARAMETER = 1.000 SCALED MAX DOF INC = -0.2176E-03
FORCE CONVERGENCE VALUE = 0.3148 CRITERION= 0.1053
EQUIL ITER 7 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -0.4987E-03
DISP CONVERGENCE VALUE = 0.4987E-03 CRITERION= 0.4290E-01 <<< CONVERGED
LINE SEARCH PARAMETER = 1.000 SCALED MAX DOF INC = -0.4987E-03
FORCE CONVERGENCE VALUE = 0.3344 CRITERION= 0.1067
EQUIL ITER 8 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.1944E-04
DISP CONVERGENCE VALUE = 0.1944E-04 CRITERION= 0.4378E-01 <<< CONVERGED
LINE SEARCH PARAMETER = 1.000 SCALED MAX DOF INC = 0.1944E-04
FORCE CONVERGENCE VALUE = 0.3273E-01 CRITERION= 0.1089 <<< CONVERGED
>>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATION 8
*** LOAD STEP 2 SUBSTEP 139 COMPLETED. CUM ITER = 1678
*** TIME = 1.98943 TIME INC = 0.105681E-01
*** AUTO STEP TIME: NEXT TIME INC = 0.10568E-01 UNCHANGED

```

[4] It uses many **Line Searches** (13.1.7, page 475). ↩



Wrap Up

[10] Save the project and exit Workbench. #

Section 13.5

Review

13.5.1 Keywords

Choose a letter for each keyword, from the list of descriptions

- | | |
|-----------------------------------|---------------------------------|
| 1. () Contact Nonlinearity | 9. () Material Nonlinearity |
| 2. () Displacement Convergence | 10. () Moment Convergence |
| 3. () Equilibrium Iterations | 11. () Newton-Raphson Method |
| 4. () Force Convergence | 12. () Nonlinear Structures |
| 5. () Geometry Nonlinearity | 13. () Residual Force |
| 6. () Linear Structures | 14. () Rotation Convergence |
| 7. () Line Search | 15. () Substeps |
| 8. () Load Steps | |

Answers:

1. (D) 2. (L) 3. (H) 4. (K) 5. (C) 6. (A) 7. (O) 8. (F) 9. (E) 10. (M)
 11. (I) 12. (B) 13. (J) 14. (N) 15. (G)

List of Descriptions

- (A) The structures in which the relation between the responses and the loads is linear.
- (B) The structures in which the relation between the responses and the loads is not linear.
- (C) Nonlinearity due to large deformation.
- (D) Nonlinearity due to the change of contact status.
- (E) Nonlinearity due to the presence of nonlinear stress-strain relation.
- (F) Also called steps. In Workbench simulations, the entire loading history can be divided into one or more load steps, so that different analysis settings can be specified for each load step.
- (G) Also called time steps. In dynamic simulations, time step size is used for integration over the time domain; the time step size must be small enough to capture the response characteristics. In static simulation, a load step can be divided into substeps small enough to achieve or enhance convergence.

- (H) Also called iterations. For nonlinear problems, each time step needs several iterations to complete. For linear problems, each time step requires exactly one equilibrium iteration.
- (I) The method used by Workbench to solve a substep in a nonlinear simulation. External force of that substep is applied. Equilibrium equation is solved for the displacement using the tangent stiffness. Internal force is calculated using updated displacement and stiffness. This completes an iteration. The process iterates until all the active convergence criteria are satisfied.
- (J) During the Newton-Raphson equilibrium iterations, the difference between external force and the calculated internal force is called a residual force.
- (K) A substep is said to satisfy the force convergence when the norm of the residual force is less than a criterion.
- (L) A substep is said to satisfy the displacement convergence when the norm of the difference of displacements between two iterations is less than a criterion.
- (M) When shell or beam elements are used in a model, moment convergence can be activated. A substep is said to satisfy the moment convergence when the norm of the residual moment is less than a criterion.
- (N) When shell or beam elements are used in a model, rotation convergence can be activated. A substep is said to satisfy the rotation convergence when the norm of the difference of rotations between two iterations is less than a criterion.
- (O) In some cases when a force-displacement relation is highly nonlinear or "concave up," during the Newton-Raphson iterations of a substep, the calculated displacement in a single iteration may overshoot the goal. In such cases, a numerical technique called line search can be activated to "scale down" the incremental displacement. Line search often helps convergence, but takes extra computing time.

13.5.2 Additional Workbench Exercises

Contact Stiffness Study

In 13.1.10[4] (page 478), when introducing **Pure Penalty** formulation, we mentioned that, in many cases, solution convergence behavior may be sensitive to **Normal Stiffness**. VM63^[Ref 1] of **Verification Manual for the Mechanical APDL Application** is a good exercise to study this behavior. First, complete a simulation using all the parameters given in the verification manual, and verify the solution with the verification manual. Next, increase the Young's modulus from the original value (1 GPa) to 200 GPa (which is the Young's modulus of steel). This should create a convergence difficulty. When you run into a convergence difficulty, try to ease the difficulty by decreasing **Normal Stiffness**. This will introduce more penetration into the solution. Tabulate data (or draw curve) to show how **Normal Stiffness** affects penetration and other results.

Reference

1. All Help>Verification Manuals>ANSYS Mechanical APDL Verification Manual>I.Verification Test Case Descriptions>VM63: Static Hertz Contact Problem

Chapter 14

Nonlinear Materials

When the stress-strain relation of a material is linear, it is called a *linear material*. On the other hand, if the stress-strain relation is not linear, it is called a *nonlinear material*. Stress-strain relation of an isotropic linear material can be expressed by Hooke's law, Eq. 1.2.8(1) (page 31), in which two independent material parameters are needed to define the stress-strain relation. So far, in foregoing simulations, we assumed the materials are linear.

In reality, most of the materials exhibit nonlinearities to some degree. In many cases, the nonlinearities are negligible, and we use Hooke's law to describe the stress-strain relation. In some other cases, when the material nonlinearities are not negligible, *nonlinear material models* must be used to define stress-strain relations. A material model is usually a mathematics form with some parameters, called *material parameters*. To assign a material to a body, you select a material model from Workbench and provide its material parameters. The material parameters are usually obtained by data-fitting using the results of a set of material testings.

Purpose of This Chapter

Workbench provides a variety of nonlinear material models. In this chapter, we will introduce two categories of nonlinear material models: *plasticity* and *hyperelasticity*. Background knowledge will be introduced first, and two step-by-step exercises will follow to demonstrate their applications.

About Each Section

Section 14.1 gives basics of plasticity and hyperelasticity. Section 14.2 provides a step-by-step example to demonstrate the use of a plastic material model. Section 14.3 uses another step-by-step example to demonstrate the application of a hyperelastic material model.

Section 14.1

Basics of Nonlinear Materials

PART A. INTRODUCTION

14.1.1 Linear versus Nonlinear Materials

[1] When the stress-strain relation of a material is linear, it is called a *linear material* [2], otherwise the material is called a *nonlinear material*. For an isotropic linear elastic material, the stress-strain relation can be expressed by Eq. 1.2.8(1) (page 31), in which two independent material parameters are needed to define the material. Note that Eq. 1.2.8(1) assumes an *isotropic material*; i.e., the Young's modulus and the Poisson's ratio are independent of directions. Orthotropic (Eq. 1.2.8(4), page 32) and anisotropic linear elasticity are also available in Workbench [3].

Besides a linear relation, Hooke's law also assumes that the stress-strain relation is elastic, time-independent, and rate-independent. Materials that violate any of these behaviors cannot be described by Hooke's law, and are categorized as nonlinear materials, even though the stress-strain relation is linear for both stressing and unloading [4].

Workbench provides non-elastic material models, called plastic material models.

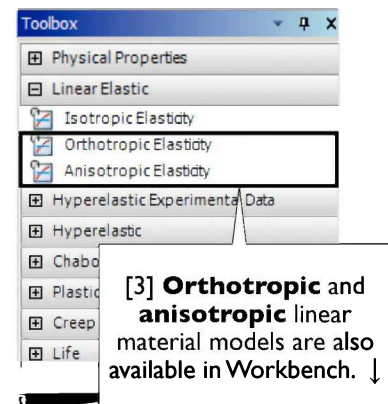
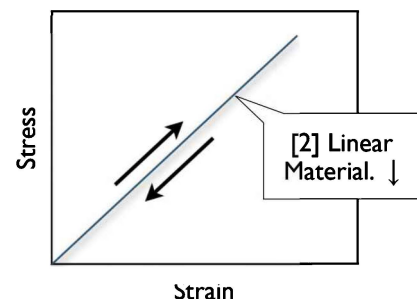
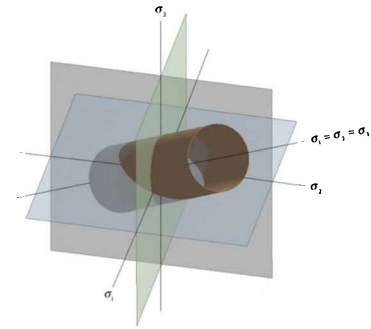
Time-Dependent Stress-Strain Relations

When you apply a stress on steel, the strain occurs instantaneously and remains the same forever. This statement is not strictly true. A more rigorous statement should be like this, "The strain occurs **ALMOST** instantaneously and **ALMOST** remains the same forever."

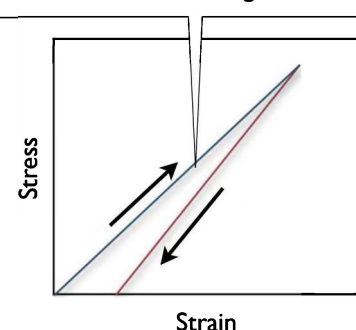
Consider a big water tank made of plastic material. Starting from the moment right after you fill up the water, the plastic tank begins to deform: the diameter expands slowly. It may take months until the deformation stops. This gives us a lesson that the deformation may take time. This time-dependent stress-strain behavior is called *creeping*, or *viscosity*.

For the steel under room temperature, the creeping behavior is practically negligible. In engineering practice, we often neglect creeping for solid materials under a fairly lower temperature, for example, below half of the melting point.

We will not discuss time-dependent behavior (although creep models are available in Workbench) for the rest of this book. ↗



[4] This is not a linear material, even though the stress-strain relation is linear for both stressing and unloading. #



14.1.2 Elastic versus Plastic Materials

[1] In 14.1.1[4] (last page), the strain is not totally recovered after release of the stress. This behavior is called plasticity, and the residual strain is called the *plastic strain*. If the strain is totally recoverable, that is, if there is no residual strain after release of the stress, the behavior is called elasticity, and the material is said to be elastic.

Following this definition of elasticity, we may classify the elastic materials into three categories: (1) linear elastic, (2) nonlinear hysteresis elastic, and (3) nonlinear non-hysteresis elastic, or simply called nonlinear elastic.

Linear Elasticity

The linear elastic material is defined by Hooke's law and depicted in 14.1.1[2], last page.

Hysteresis Elasticity

The term *elastic hysteresis* has been introduced in 12.1.3[4-7] (page 424). The current version of ANSYS (even with APDL) doesn't directly provide a material model to include the hysteresis behavior. However, you may include the hysteresis behavior in terms of material damping (12.1.3, page 423-425).

Most materials have hysteresis behavior to some extent. However, as long as it is small enough, we may neglect the hysteresis behavior.

Nonlinear Elasticity

Nonlinear non-hysteresis elastic materials are characterized by the fact that the stressing curve and the unstressing curve are coincident [2]: the energy is conserved in stressing-unstressing cycles.

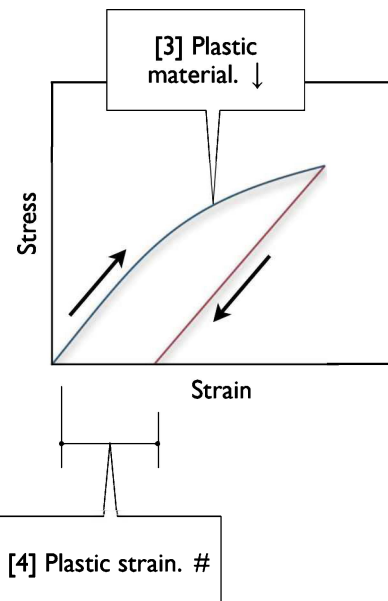
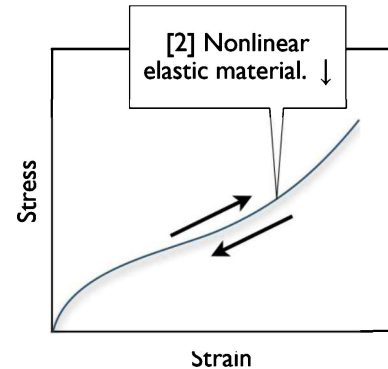
The challenge of implementing nonlinear elastic material models is that the strain may be as large as 100% or even 200%, such as rubber under stretching or compression. Under such large strains, the stretching and compression behaviors may not be described by the same parameters. This kind of super-large deformation elasticity is given a special name: *hyperelasticity*.

Workbench provides many hyperelastic material models and we will discuss them in PART C of this section (pages 532-535). A step-by-step example is given in Section 14.3.

Plasticity

Plastic materials are characterized by the presence of the residual strain, or plastic strain [3-4]. Note that the hysteresis is always present in plastic materials: there is always energy loss in stressing-unstressing cycles.

Workbench provides many plastic material models and we will discuss them in PART B of this section (starting from next page). A step-by-step example is given in Section 14.2. ↗



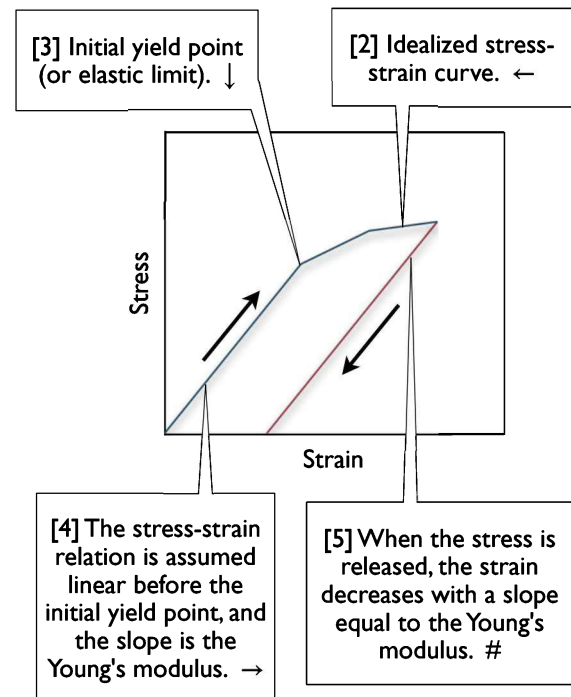
PART B. PLASTICITY

14.1.3 Idealized Stress-Strain Curve for Plasticity

[1] Plasticity behavior typically occurs in ductile metals subject to large deformation. Plastic deformation is a result of slips between grains due to shear stresses. It is essentially a rearrangement of atoms in the crystal structure.

In Workbench, a typical stress-strain relation, such as 14.1.2[3] (last page), is idealized as shown in [2-5]. The stress-strain curve is composed of several straight segments. The slope of the first segment is the Young's modulus [4]. When the stress is released, the strain decreases with a slope equal to the Young's modulus [5]. This implies that if the stress/strain state is on the first segment, the behavior is elastic and no plastic strain remains after releasing the stress. The point at the end of the first segment is called *elastic limit*, or *initial yield point* [3]. All points higher than the initial yield point are called *subsequent yield points*, since they all represent *yield states*.

A uniaxial stress-strain relation such as [2-5] is not sufficient to fully define a plasticity behavior. There are other characteristics that must be described for general multiaxial cases: (a) What is the yield criterion? (b) What is the hardening rule? ↗



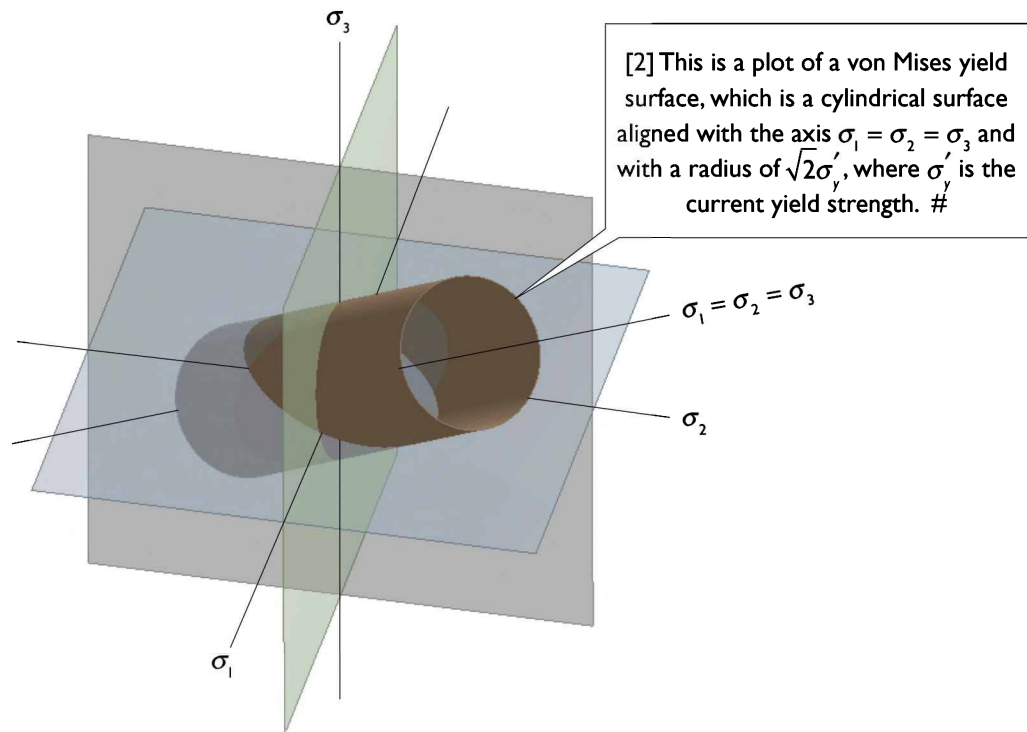
14.1.4 Yield Criteria

[1] A stress-strain curve such as 14.1.3[2-5] is usually obtained by a uniaxial tensile test. It provides an initial yield strength σ_y of the uniaxial tensile test. In three-dimensional cases, the stress state is multiaxial. According to what criteria can we say that a stress state reaches a yield state? Workbench uses von Mises criterion (1.4.5, pages 43-46) as the yield criterion; i.e., a stress state reaches yield state when the von Mises stress σ_e is equal to the CURRENT uniaxial yield strength σ'_y , or

$$\sqrt{\frac{1}{2}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]} = \sigma'_y \quad (1)$$

The yielding initially occurs when $\sigma'_y = \sigma_y$, and the "current" uniaxial yield strength σ'_y may change subsequently. As mentioned in 1.4.5[4] (page 46), when plotted in the σ_1 - σ_2 - σ_3 space, Eq. (1) is a cylindrical surface aligned with the axis $\sigma_1 = \sigma_2 = \sigma_3$ and with a radius of $\sqrt{2}\sigma'_y$. It is called a *von Mises yield surface* ([2], next page). If the stress state is inside the cylinder, no yielding occurs. If the stress state is on the surface, yielding occurs. No stress state can be outside the yield surface. If the stress state is on the surface and the loads continue to "push" the yield surface outward, the size (radius) or the location of the yield surface will change. The rule that describes how the yield surface changes its size or location is called a *hardening rule* (14.1.5, next page).

Note that, in a uniaxial test, we are talking about "yield points" in the stress axis. In a biaxial case, the yielding states form a "yield line" in a stress plane, while in 3D cases, the yielding states become a "yield surface" in a stress space ([2], next page). ↙



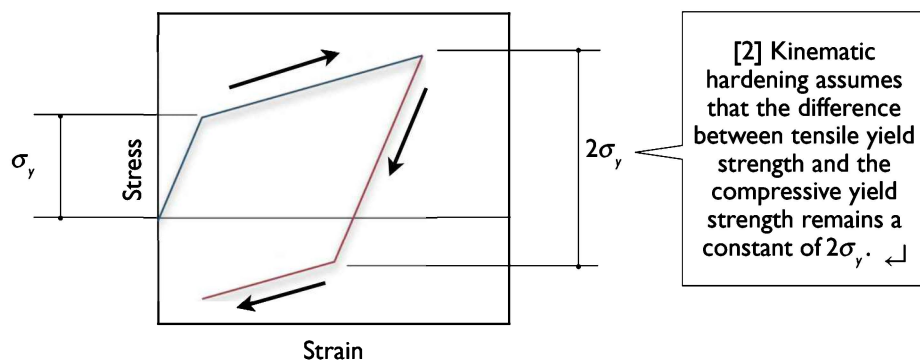
14.1.5 Hardening Rules

[1] Workbench implements two hardening rules: (a) kinematic hardening, and (b) isotropic hardening. In metal plasticity, hardening behavior is often a mix-up of kinematic and isotropic.

Kinematic Hardening

Kinematic hardening assumes that, if a stress state is on the yield surface and the loads continue to "push" a yield surface outward, the yield surface will change its location, according to the "pushing direction," but preserve the size of the yield surface. In a uniaxial test, it is equivalent to say that the difference between the tensile yield strength and the compressive yield strength remains a constant of $2\sigma_y$ [2].

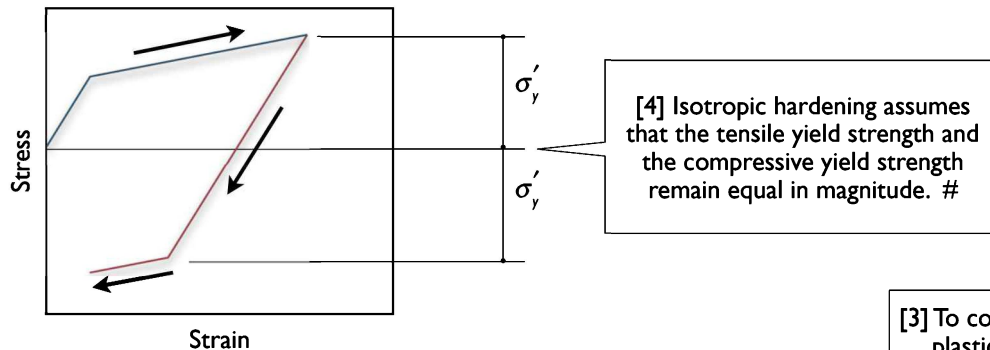
Kinematic hardening is generally used for small strain, cyclic loading applications, especially metals. ↓



Isotropic Hardening

[3] Isotropic hardening assumes that, when the loads continue to "push" a yield surface, the yield surface will expand its size, but preserve the axis of the yield surface. In a uniaxial test, it is equivalent to say that the "current" tensile yield strength and the compressive yield strength remain equal in magnitude [4].

Isotropic hardening is often used for large strain simulations. It is usually not applicable for cyclic loading applications. ↘



14.1.6 Workbench Plasticity Models

[1] Workbench provides many plasticity models [2]. Besides choosing from either of the hardening rules, you can choose from either type of stress-strain curves: bilinear or multilinear.

Linear Elastic Properties Must be Included

To describe a material model for plasticity, you must include a set of linear elastic properties (e.g., Young's modulus and Poisson's ratio in cases of isotropy). The Young's modulus is used as the initial slope of the stress-strain curve, in which the material initially behaves as linear elasticity.

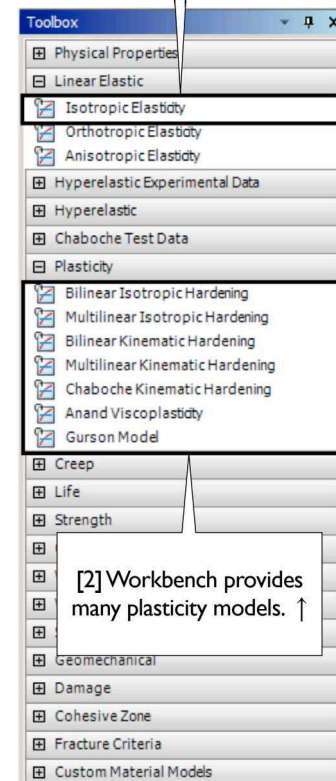
Bilinear Stress-Strain Curve

Examples of bilinear models are shown in 14.1.5[2, 4] (last and this pages); the stress-strain curve is composed of two straight segments. Besides the Young's modulus and Poisson's ratio, you need to supply an *initial yield stress* and a *tangent modulus* (the slope of the second segment).

Multilinear Stress-Strain Curve

An example of multilinear models was shown in 14.1.3[2-5] (page 529); the stress-strain curve is composed of several straight segments. Besides the Young's modulus and Poisson's ratio, you need to supply a tabular form of data describing the *subsequent yield stresses* and the corresponding *plastic strains*. An example is given in 14.2.6[6-10], pages 548-549. →

[3] To complete a description of plasticity model, you must include its linear elastic properties (e.g., Young's modulus and Poisson's ratio for isotropic materials). #



PART C. HYPERELASTICITY

14.1.7 Test Data Needed for Hyperelasticity

[1] As mentioned in 14.1.2[1] (page 528), the challenge of implementing nonlinear elastic models is that the strain may be as large as 100% (or even 200%), such as rubber under stretching.

In plasticity or linear elasticity, we use a stress-strain curve to describe its behavior, and the stress-strain curve is usually obtained by a uniaxial tensile test. Since only tension behavior is investigated, other behaviors (e.g., compressive, shearing) must be drawn from the tensile test data. In plasticity or linear elasticity, we implicitly made the following assumptions: (a) The compressive behavior is symmetric to the tension behavior; i.e., they have the same Young's modulus, and the same Poisson's ratio. The symmetry may not be true when the strain is large. We may need to conduct a compressive test to assess the compressive behavior. (b) The shear modulus G is related to the Young's modulus and the Poisson's ratio by Eq. 1.2.8(2) (page 31). Again, this assumption may not be true when the strain is large. We may need to conduct a shear test to assess the shearing behavior. (c) We also assume that the bulk modulus B is related to the Young's modulus and the Poisson's ratio by

$$B = \frac{E}{3(1-2\nu)} \quad (1)$$

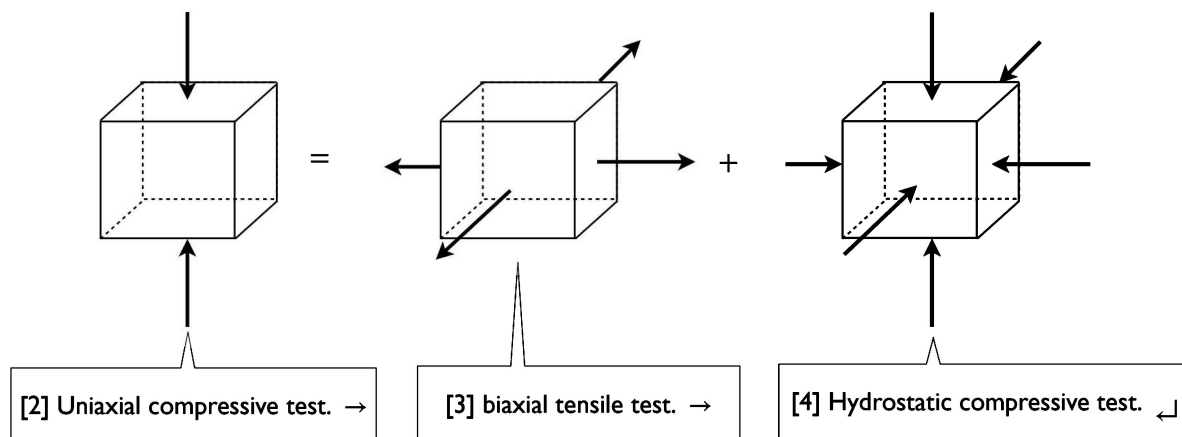
Again, this assumption may not be true when the strain is large. We may need to conduct a volumetric test to assess the volumetric behavior. However, in many cases, when the bulk modulus is almost infinitely large (i.e., the material is incompressible), we usually assume incompressibility without conducting a volumetric test.

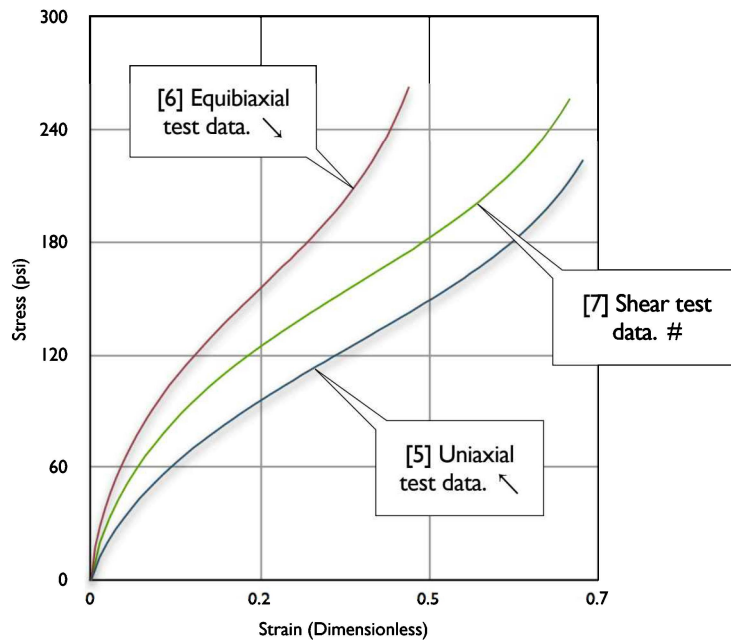
Further, when the strain is large, all the moduli (tensile, compressive, shear, and bulk) are no longer constant; they change along stress-strain curves. Nonlinear elasticity with large strain is also called *hyperelasticity*.

In summary, to describe hyperelasticity behavior, we may need the following test data: (a) a set of uniaxial tensile test data, (b) a set of uniaxial compressive test data, (c) a set of shear test data, and (d) a set of volumetric test data if the material is compressible.

Often, a set of test data can be obtained by superposing two sets of other test data. For example, the set of uniaxial compressive test data can be obtained by adding a set of hydrostatic compressive test data to a set of equibiaxial tensile test data [2-4]. There are two reasons for doing this: (a) Biaxial tensile test may be easier to conduct than compressive test in some testing devices; (b) For incompressible materials, hydrostatic compressive test data are trivial: all strains have zero values.

An example of test data for hyperelasticity is shown in [5-7] (next page), which will be used in Section 14.3. ✓





14.1.8 Strain Energy Function

[1] Workbench provides a material model, called **Response Function** (see 14.1.9[2], page 535), which uses experimental data (such as 14.1.7[5-7], this page) directly. A drawback of using **Response Function** model is that it may not be efficient enough--it may cost too many iterations, even if convergence is eventually achieved.

A better idea is described as follows.

As mentioned in 14.1.2[1] (page 528), hyperelasticity is characterized by the fact that the stressing curve and the unstressing curve are coincident (14.1.2[2], page 528). During the stressing and unstressing, the energy is conserved, or, in other words, the stressing and unstressing are path independent. The stress state depends only on the strain state, and vice versa. They are independent of the stressing/unstressing history. This implies that there exists a *potential energy function* that depends on the state of the stress or strain. It reminds us of the strain energy density function, which does depend only on the state of stress or strain. With these in mind, we propose a mathematical form for the strain energy

$$W = W(\varepsilon_{ij}) \quad (1)$$

And the stress can be calculated from the strain energy using

$$\sigma_{ij} = \frac{\partial W}{\partial \varepsilon_{ij}} \quad (2)$$

The strain state ε_{ij} consists of 6 strain components (Eq. 1.2.4(4), page 28). To further simplify the strain energy function and develop a coordinate-independent expression, we may replace the 6 strain components (which are coordinate-dependent) with 3 strain invariants (which are coordinate-independent). Before going further, we need more background. Let's introduce some terms in solid mechanics. ↵

Principal Stretch Ratios

[2] The *stretch ratio* λ is defined as the ratio between fiber lengths after and before deformation,

$$\lambda = \frac{L}{L_0} = 1 + \varepsilon \quad (3)$$

When the direction is along a principal direction, the stretch ratio is called a *principal stretch ratio*. There are 3 principal stretch ratios, denoted by λ_1 , λ_2 , and λ_3 , which provide a measure of the deformation.

The *volumetric ratio* J can be defined as the volume after and before deformation,

$$J = \frac{V}{V_0} = \lambda_1 \lambda_2 \lambda_3 \quad (4)$$

Note that, if the material is incompressible, $J = 1$.

The *deviatoric principal stretch ratios* are defined as

$$\begin{aligned} \bar{\lambda}_1 &= \lambda_1 / \sqrt[3]{J} \\ \bar{\lambda}_2 &= \lambda_2 / \sqrt[3]{J} \\ \bar{\lambda}_3 &= \lambda_3 / \sqrt[3]{J} \end{aligned} \quad (5)$$

Strain Invariants

Let I_1 , I_2 , and I_3 be the characteristic values (eigenvalues) of the strain state; they are also called *strain invariants*. It can be proved that

$$\begin{aligned} I_1 &= \lambda_1^2 + \lambda_2^2 + \lambda_3^2 \\ I_2 &= \lambda_1^2 \lambda_2^2 + \lambda_2^2 \lambda_3^2 + \lambda_3^2 \lambda_1^2 \\ I_3 &= \lambda_1^2 \lambda_2^2 \lambda_3^2 \end{aligned} \quad (6)$$

The *deviatoric strain invariants* are defined as

$$\begin{aligned} \bar{I}_1 &= \lambda_1 / \sqrt[3]{J^2} \\ \bar{I}_2 &= \lambda_2 / \sqrt[3]{J^2} \\ \bar{I}_3 &= \lambda_3 / \sqrt[3]{J^2} \end{aligned} \quad (7)$$

Strain Energy Functions

We can replace the 6 strain components in Eq. (1) with either strain invariants or principal stretch ratios; i.e.,

$$W = W(I_1, I_2, I_3)$$

or

$$W = W(\lambda_1, \lambda_2, \lambda_3)$$

Or, we can split the strain energy into deviatoric part and volumetric part, and write

$$W = W_d(\bar{I}_1, \bar{I}_2) + W_b(J) \quad (8)$$

or

$$W = W_d(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3) + W_b(J) \quad (9)$$

Note that $I_3 = J^2$, so \bar{I}_3 is not used in the definition of W . #

14.1.9 Workbench Hyperelasticity Models

[1] Workbench provides many hyperelasticity models [2]; they are based on either Eq. 14.1.8(8) or Eq. 14.1.8(9), last page. ↘

Polynomial Form

[3] The polynomial form is based on Eq. 14.1.8(8), last page,

$$W = \sum_{i+j=1}^N c_{ij} (\bar{I} - 3)^i (\bar{I}_2 - 3)^j + \sum_{k=1}^N \frac{1}{d_k} (J - 1)^{2k} \quad (1)$$

For example, **Polynomial 1st Order** ($N = 1$) is

$$W = c_{10} (\bar{I} - 3) + c_{01} (\bar{I}_2 - 3) + \frac{1}{d_1} (J - 1)^2 \quad (2)$$

It has three parameters, c_{10} , c_{01} , and d_1 . Note that, for incompressible materials, $J = 1$, $d_1 = 0$, and the last term is dropped.

Ogden Form

The Ogden form is based on Eq. 14.1.8(9), last page,

$$W = \sum_{i=1}^N \frac{\mu_i}{\alpha_i} (\bar{\lambda}_1^{\alpha_i} + \bar{\lambda}_2^{\alpha_i} + \bar{\lambda}_3^{\alpha_i} - 3) + \sum_{i=1}^N \frac{1}{d_i} (J - 1)^{2i} \quad (3)$$

For example, **Ogden 1st Order** ($N = 1$) is

$$W = \frac{\mu_1}{\alpha_1} (\bar{\lambda}_1^{\alpha_1} + \bar{\lambda}_2^{\alpha_1} + \bar{\lambda}_3^{\alpha_1} - 3) + \frac{1}{d_1} (J - 1)^2 \quad (4)$$

It has three parameters, μ_1 , α_1 , and d_1 .

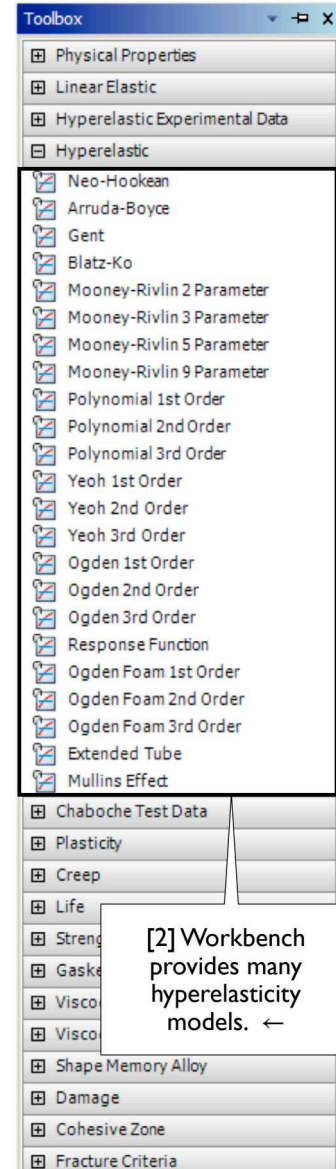
Mooney-Rivlin, Yeoh, and Neo-Hookean

These are reduced forms of the generalized polynomial.

Which Form to Use?

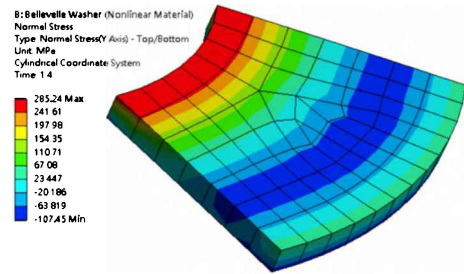
The choice depends on type of material, maximum strain, and the test data available. In general, the best form of strain energy density function is the one that produces the closest curve fit of the test data.

Section 14.3 provides an example to demonstrate some details. #



Section 14.2

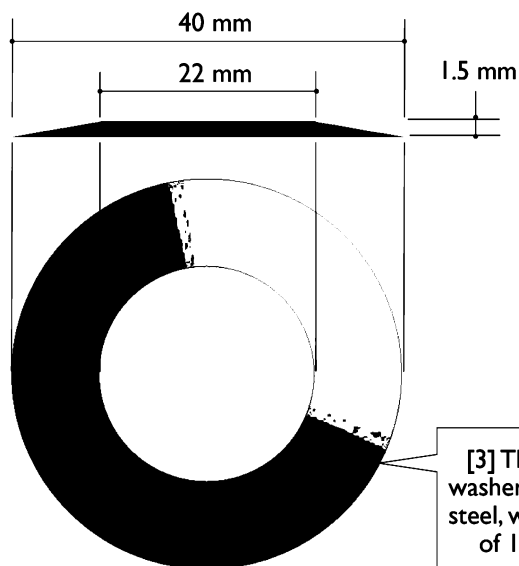
Belleville Washer



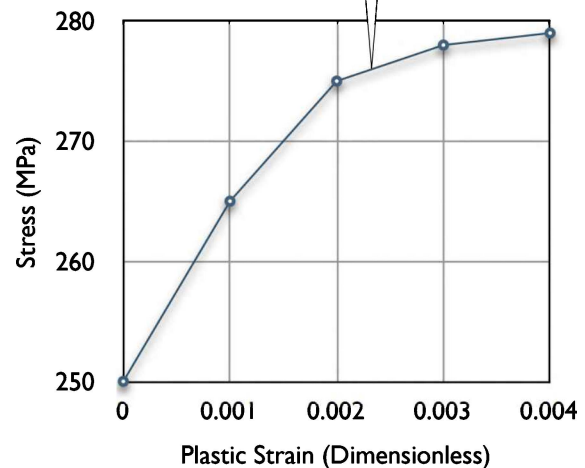
14.2.1 About the Belleville Washer

[1] The dimensions of a *Belleville washer* [2], also called a *Belleville spring*, are shown in [3]. The washer is made of a steel with a Young's modulus of 200 GPa, a Poisson's ratio of 0.3, and an initial yield stress of 250 MPa. Beyond the initial yield stress, it displays plasticity behavior as shown in [4].

In this section, we will compress the Belleville spring by 1.0 mm and then release the displacement completely. As a preliminary study, before performing a simulation with plasticity, we will assume a linear material to see if the maximum stress exceeds the yield stress. If so, then we will explore the plasticity behavior of the Belleville spring. Specifically, we will examine the residual stress after the spring is completely released. A force-displacement curve will be plotted. ↗

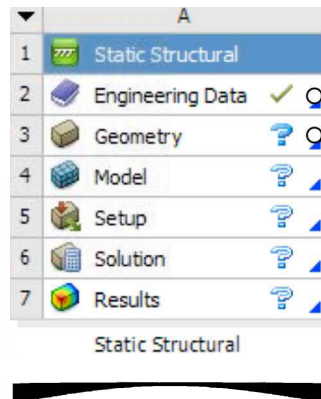


[4] The stress-versus-plastic-strain curve of the steel in this case. (For the tabular data, see 14.2.6[9], page 549.) #



14.2.2 Start Up

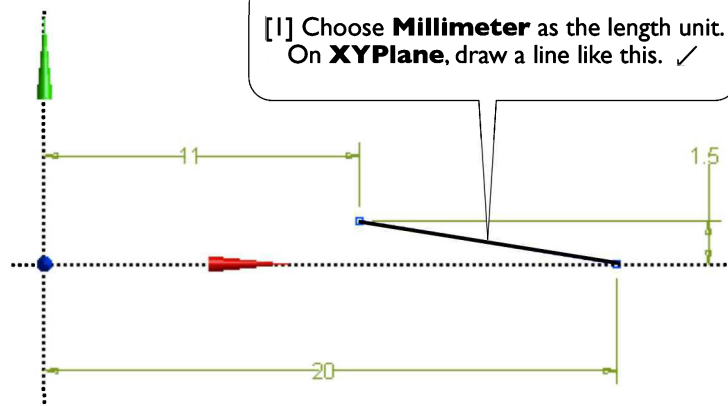
[1] Launch Workbench. Create a **Static Structural** analysis system. Save the project as **Belleville**. ↓



[2] Assuming a linear material for the structural steel, we don't need to do anything with **Engineering Data** for now, since **Isotropic Elasticity** is the default material model for the structural steel; its Young's modulus is 200 GPa and Poisson's ratio is 0.3. ↓

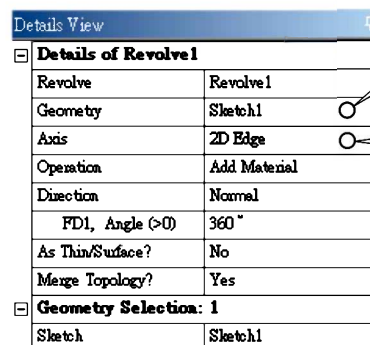
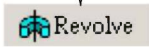
[3] Start up DesignModeler. #

14.2.3 Create Geometry



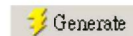
[1] Choose **Millimeter** as the length unit. On **XYPlane**, draw a line like this. ✓

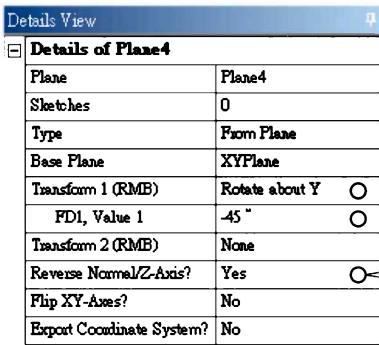
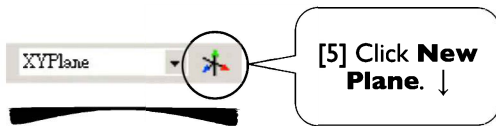
[2] Click **Revolve**. →



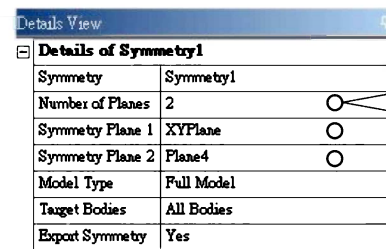
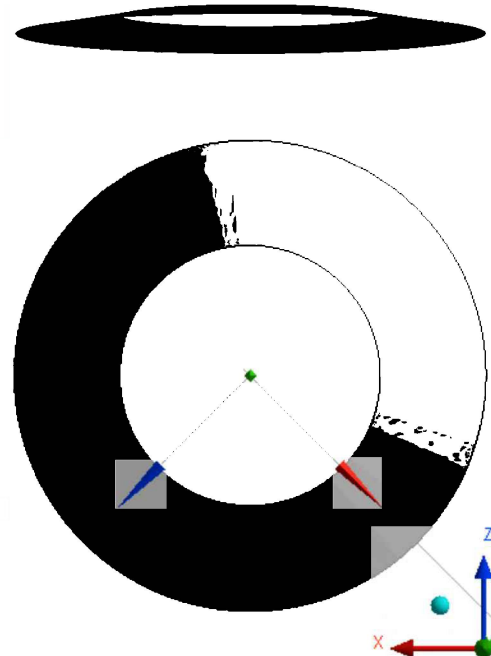
[3] Click **Apply**. ↓

[4] Select the Y-axis as the axis of revolution. ↵





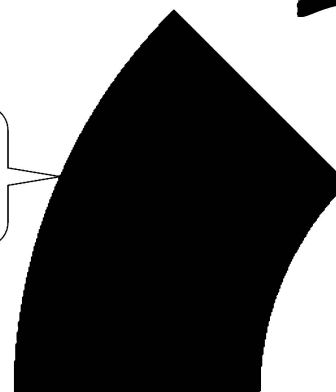
[6] We want to use this plane as a plane of symmetry. Remember that, when using the **Symmetry** tool [7], the material to the -Z side is removed. ↓



[8] Two planes of symmetry. ✓

[7] Select **Tools/ Symmetry**. →

[9] Now, only 1/8 of the model remains. Close DesignModeler. ↵



[10] There are actually infinite planes of symmetry for this mode (any plane derived from rotating **XYPlane** about Y-axis could be a plane of symmetry for this model). In general, an axisymmetric geometry has an infinite number of planes of symmetry; in 2D, we can use any two of them. In this case, we decide to use 1/8 of the model. In practice, we often choose a 45° sector of the model. Alternatively, we may model the problem as an axisymmetric problem using 2D option. This is left as an exercise (14.4.2, page 569). #

14.2.4 Set Up for Simulation with Linear Material

[1] Start up **Mechanical**. Make sure the unit system is **mm-kg-N-s**. ←

[2] Highlight **Surface Body**. ↓

[3] Type **1 (mm)**. ↓

[4] **Structural Steel** is used by default. ↗

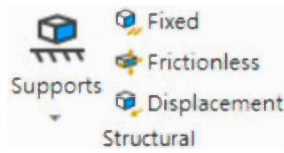
[5] Highlight **Analysis Settings**. ↓

[6] Type **2** for **Number of Steps**. ↓

[7] Turn on **Large Deflection**. ↵

Graphics Properties	
Definition	
Suppressed	No
Dimension	3D
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Thickness	1. mm
Thickness Mode	Manual
Offset Type	Middle
Treatment	None
Material	
Assignment	Structural Steel
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	
Statistics	
CAD Attributes	
DMSheetThickness	0

Step Controls	
Number Of Steps	2
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Restart Controls	
Nonlinear Controls	
Output Controls	
Analysis Data Management	



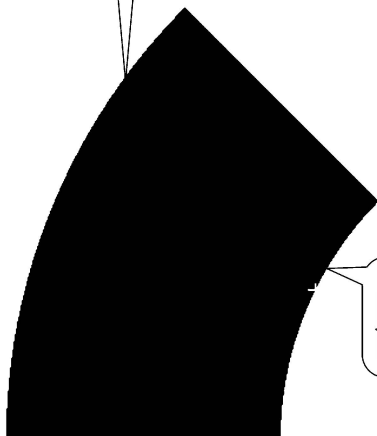
[8, 12] Insert a **Displacement**. ✓ ↓

Details of "Displacement"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
<input checked="" type="checkbox"/> Y Component	0. mm (ramped)
Z Component	Free
Suppressed	No

[10] Click **Apply**. ↓

[11] Fix the edge in Y-direction. ←

[9] Select this edge (using the edge filter). ↗



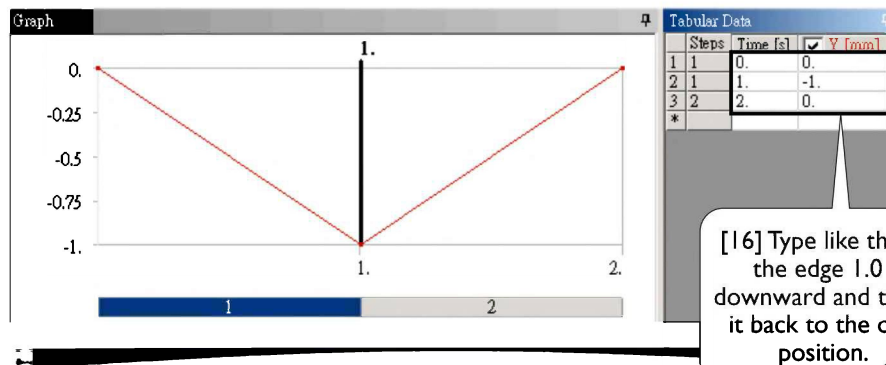
[13] Select this edge. ↗



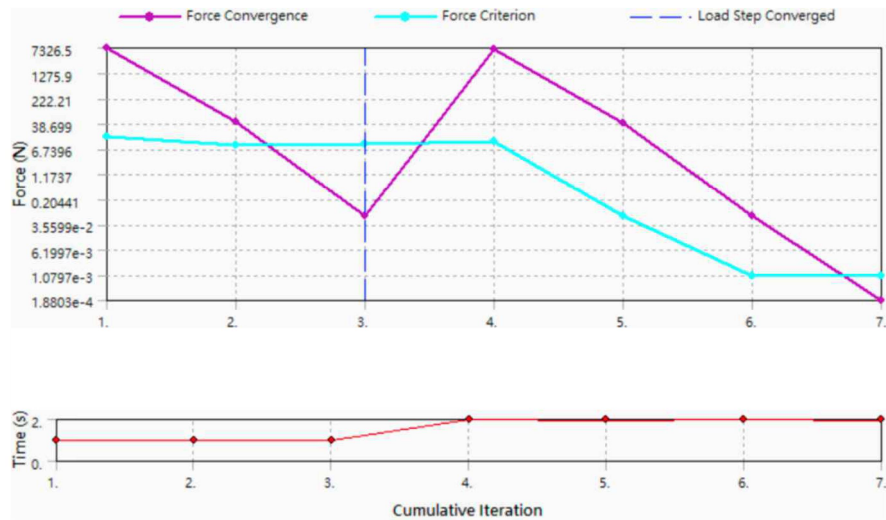
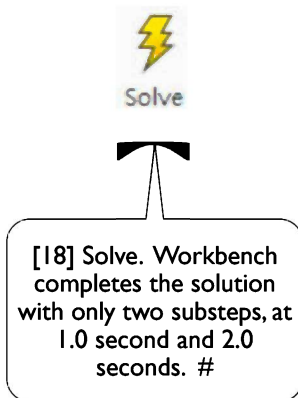
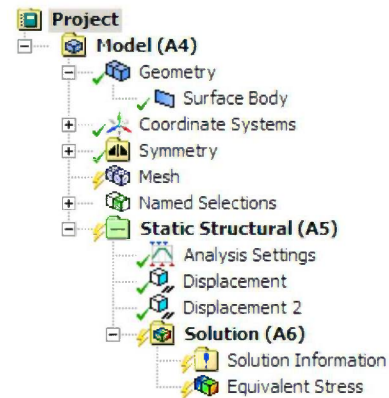
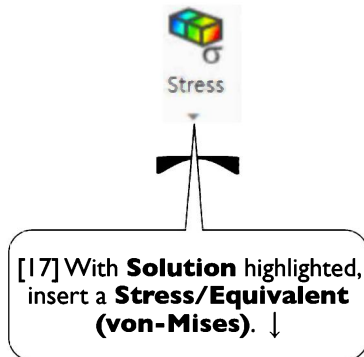
Details of "Displacement 2"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
<input checked="" type="checkbox"/> Y Component	Tabular Data
Z Component	Free
Suppressed	No
Tabular Data	
Independent Variable	Time

[14] Click **Apply**. ↓

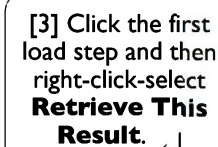
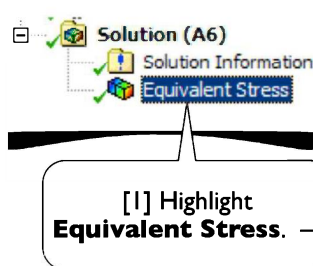
[15] Select **Tabular** for Y Component. ↓



[16] Type like this: push the edge 1.0 mm downward and then pull it back to the original position. ↵



14.2.5 Results of the Linear Material Simulation



Tabular Data			
Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1. 1.	81.407	1955.6	718.4
2. 2.	0.26009	3.5098	1.0597

[2] **Tabular Data** shows that, at the end of first load step, the maximum stress is 1955.6 MPa and, at the end of the second load step, the maximum stress is essentially zero (it must be so for an elastic material). ←

A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress - Top/Bottom
Unit: MPa
Time: 1

1955.6 Max
1747.3
1539.1
1330.8
1122.6
914.37
706.13
497.89
289.65
81.407 Min

Named Selection
Coordinate System
Remote Point

[6] Click **Coordinate System** and rename the new coordinate system as **Cylindrical Coordinate System**.

[4] Most of the area has stress larger than the yield stress (250 MPa, see 14.2.1 [4], page 536). This is not realistic. However, to compare the results with those in the model using a plastic material, let's explore some other behaviors. ✓

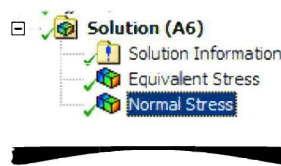
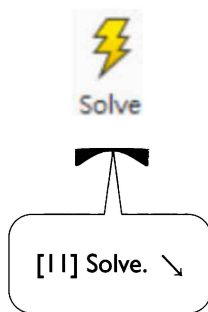
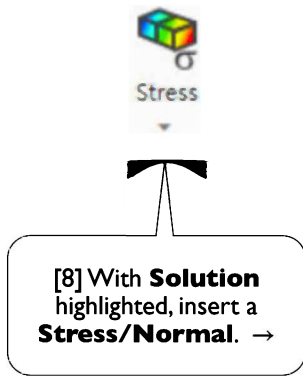
[5] Highlight **Coordinate Systems**.

[7] Set up the coordinate system like this.

Project*
Model (A4)
Geometry
Surface Body
Materials
Coordinate Systems
Global Coordinate System
XYPlane
Plane4
Cylindrical Coordinate System
Symmetry
Mesh
Named Selections
Static Structural (A5)
Analysis Settings
Displacement
Displacement 2
Solution (A6)
Solution Information
Equivalent Stress

Details of "Cylindrical Coordinate System"

Definition	
Type	Cylindrical
Coordinate System	Program Controlled
APDL Name	
Suppressed	No
Origin	
Define By	Global Coordinates
Origin X	0. mm
Origin Y	0. mm
Origin Z	0. mm
Location	Click to Change
Principal Axis	
Axis	X
Define By	Global X Axis
Orientation About Principal Axis	
Axis	Z
Define By	Global Y Axis
Directional Vectors	
Transformations	
Base Configuration	Absolute
Transformed Configuration	[0. 0. 0.]

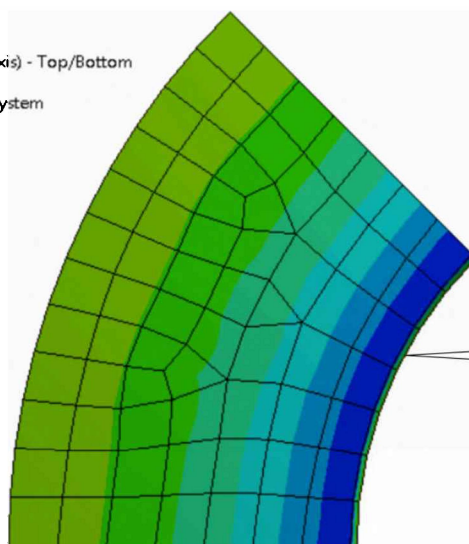
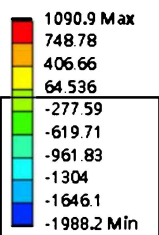


[9] Select **Cylindrical Coordinate System** and select **Y Axis**. Now, Y-direction is read θ -direction, or tangential direction. This stress is also called the hoop stress. ↓

Details of "Normal Stress"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Position	Top/Bottom
Definition	
Type	Normal Stress
Orientation	Y Axis
By	Time
Display Time	1. s
Coordinate System	Cylindrical Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
Results	
Minimum	
Maximum	
Minimum Occurs On	
Maximum Occurs On	
Information	

[10] Display the hoop stress at the end of the first load step. ←

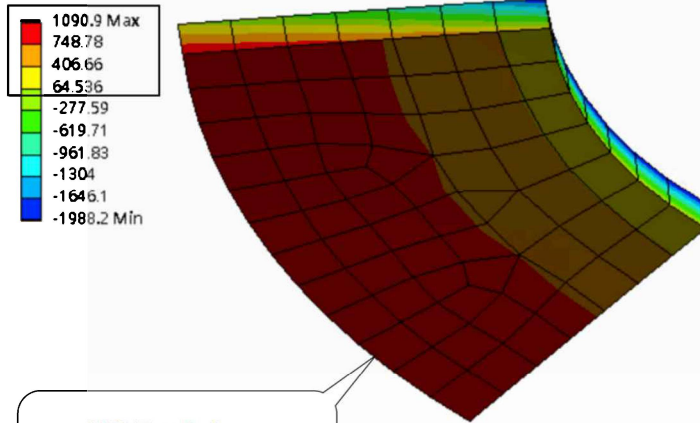
A: Static Structural
Normal Stress
Type: Normal Stress(Y Axis) - Top/Bottom
Unit: MPa
Cylindrical Coordinate System
Time: 1



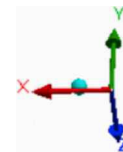
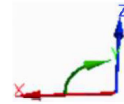
[12] Highlight **Normal Stress**. Compressive hoop (tangential) stress dominates the top surface. ↙



A: Static Structural
Normal Stress
Type: Normal Stress(Y Axis) - Top/Bottom
Unit: MPa
Cylindrical Coordinate System
Time: 1



[13] Tensile hoop (tangential) stress dominates the bottom surface. ↓



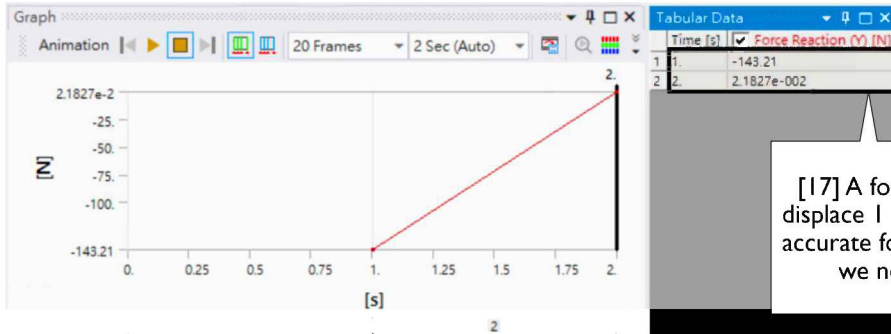
[14] With **Solution** highlighted, insert a **Probe/Force Reaction**. →

Details of "Force Reaction"	
[-] Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Displacement 2
Orientation	Global Coordinate System
Suppressed	No
[-] Options	
Result Selection	Y Axis
Display Time	End Time
[+] Results	
[-] Maximum Value Over Time	
Y Axis	
[-] Minimum Value Over Time	
Y Axis	
[+] Information	

[15] We want to know the force required to displace the Belleville spring. ↓



[16] Solve. ↩



[17] A force of 143.21 N is required to displace 1 mm downward. To plot a more accurate force-versus-displacement curve, we need smaller time steps. ✓

Details of "Analysis Settings"

Step Controls		
Number Of Steps	2.	
Current Step Number	1.	
Step End Time	1. s	
Auto Time Stepping	Off	
Define By	Time	
Time Step	0.1 s	
Solver Controls		
Solver Type	Program Controlled	
Weak Springs	Off	
Solver Pivot Checking	Program Controlled	
Large Deflection	On	
Inertia Relief	Off	
Restart Controls		
Nonlinear Controls		
Output Controls		
Analysis Data Management		
Visibility		

[18] Set up **Analysis Settings** for the first load step. →

Details of "Analysis Settings"

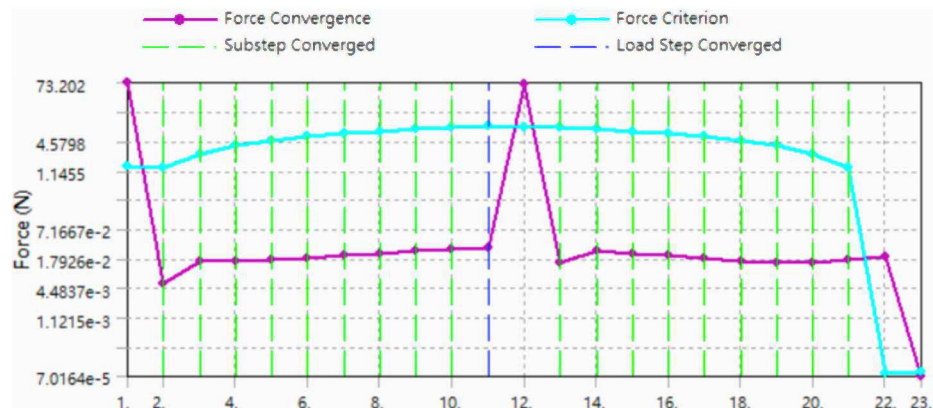
Step Controls		
Number Of Steps	2.	
Current Step Number	2.	
Step End Time	2. s	
Auto Time Stepping	Off	
Define By	Time	
Time Step	0.1 s	
Solver Controls		
Solver Type	Program Controlled	
Weak Springs	Off	
Solver Pivot Checking	Program Controlled	
Large Deflection	On	
Inertia Relief	Off	
Restart Controls		
Nonlinear Controls		
Output Controls		
Analysis Data Management		
Visibility		

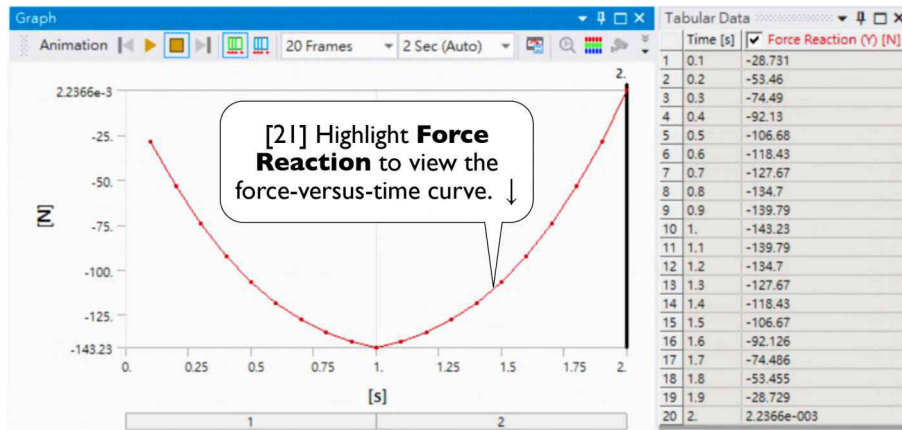
[19] Set up **Analysis Settings** for the second load step. ✓



Solve

[20] Solve. ↩





[22] With **Solution** highlighted, click **User Defined Result**. ↓

Details of "User Defined Result"

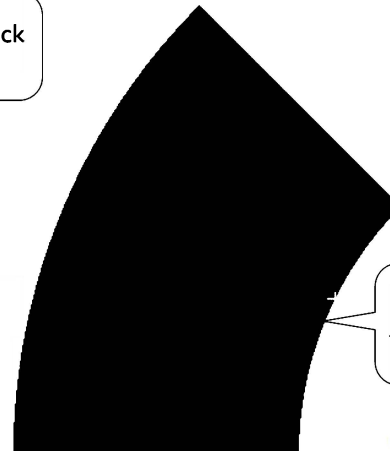
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Sub Scope By	Layer
Layer	Entire Section
Position	Top/Bottom
Definition	
Type	User Defined Result
Expression	= UY
Input Unit System	Metric (mm, kg, N, s, mV, mÅ)
Output Unit	
By	Time
Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
Results	
Minimum	
Maximum	
Information	

[24] Click **Apply**. ↓

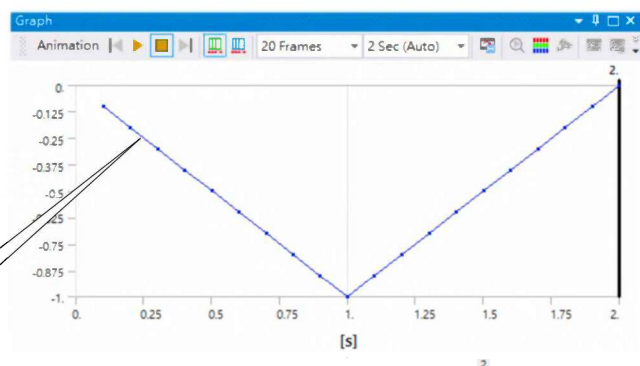
[25] Type **UY** (without an equal sign). ↓



[26] Solve. ↓

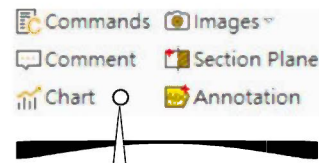


[23] Select this edge. ←



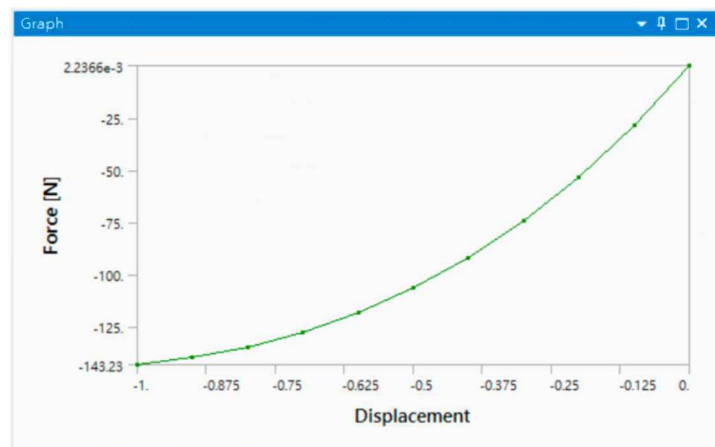
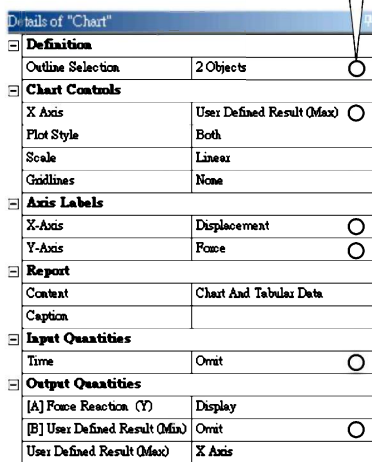
Create a Force-versus-Displacement Curve

[28] In step [21] (last page), we have a force-versus-time curve, while in step [27] we have a displacement-versus-time curve. Now, we want to combine them and create a force-versus-displacement curve. →

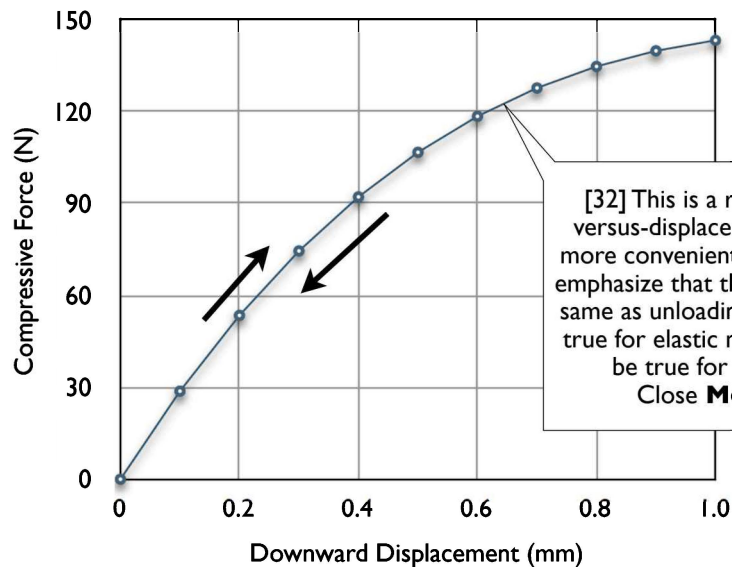


[29] Click **Chart**. ←

[30] Click to bring up **Apply/Cancel** buttons and control-select **Force Reaction** and **User Defined Result** in the project tree and click **Apply**. ↓



[31] A force-versus-displacement curve. ↓



[32] This is a re-plot of the force-versus-displacement curve [31] in a more convenient fashion. We want to emphasize that the loading curve is the same as unloading curve; this is always true for elastic materials, but may not be true for plastic materials.
Close **Mechanical**. #

14.2.6 Set Up for Simulation with Plastic Material Model

[1] Right-click here and select **Duplicate**. ✓

[2] Rename like this for the old system. ↓

[3] Rename like this for the new system. ↗

[4] Double-click **Engineering Data** of the new system. ↘

[5] Select **tonne-mm-s** as project units and select **Display Values in Project Units**. ✓

[6] Expand **Plasticity** and double-click **Multilinear Isotropic Hardening** to include it in **Structural Steel**. ↶

The image shows a software interface with a tree view on the left and a properties table on the right. The tree view shows a hierarchy of systems: A (Static Structural), B (Engineering Data), C (Geometry), D (Model), E (Setup), F (Solution), and G (Results). The properties table shows the material properties for 'Structural Steel'.

Property	Value	Unit
Density	7.85E-09	tonne mm ⁻³
Isotropic Secant Coefficient of Thermal Expansion		
Isotropic Elasticity		
Derive from	Young's Modulus and Poisson's Ratio	
Young's Modulus	2E+05	MPa
Poisson's Ratio	0.3	
Bulk Modulus	1.6667E+05	MPa
Shear Modulus	76923	MPa
Alternating Stress Mean Stress		
Strain-Life Parameters		
Tensile Yield Strength	250	MPa
Compressive Yield Strength	250	MPa
Tensile Ultimate Strength	460	MPa
Compressive Ultimate Strength	0	MPa

[11] Return to **Project Schematic**.

[8] Type 22 (degree C) for the temperature. →

[9] Type these data (see 14.2.1[4], page 536). Make sure the stress unit is MPa. ↓

[7] Highlight **Tabular of Multilinear Isotropic Hardening**. ↑

[10] The stress-versus-plastic-strain curve.

[12] Start up **Mechanical**. ✓

[13] Highlight **Solution Information**. →

[14] Solve. ↩

Plastic Strain (mm mm ⁻¹)	Stress (MPa)
0	250
0.001	265
0.002	275
0.003	278
0.004	279

Chart of Properties Row 12: Multilinear Isotropic Hardening

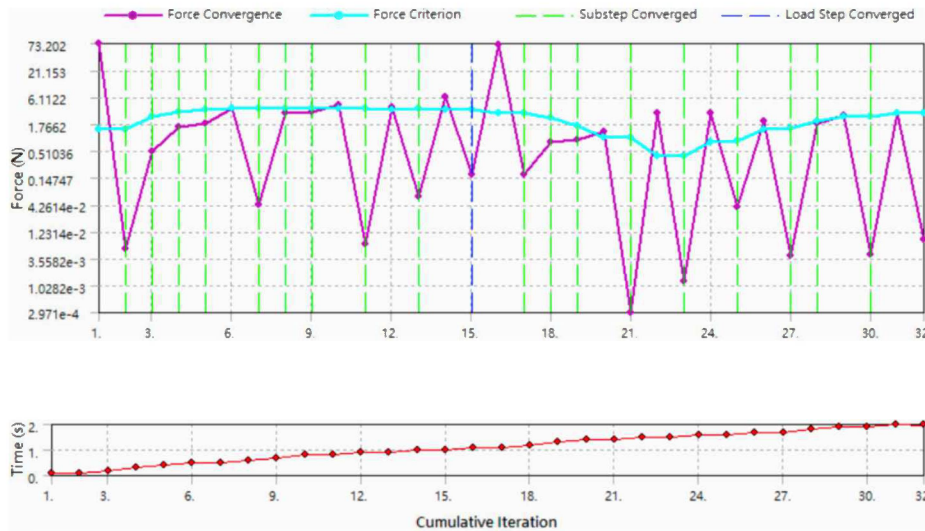
Stress (MPa)

Plastic Strain [mm mm⁻¹]

Temperature: 22 [C]

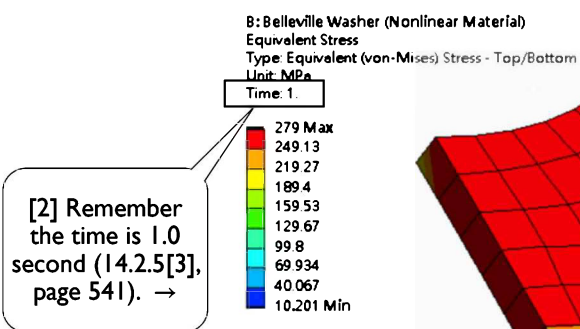
Belleville Washer (Nonlinear Material)

Solve

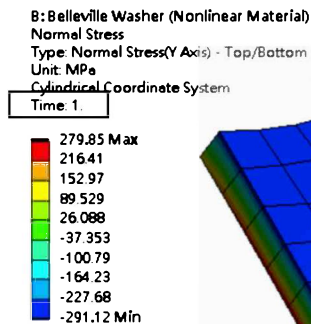


[15] The force convergence curves. #

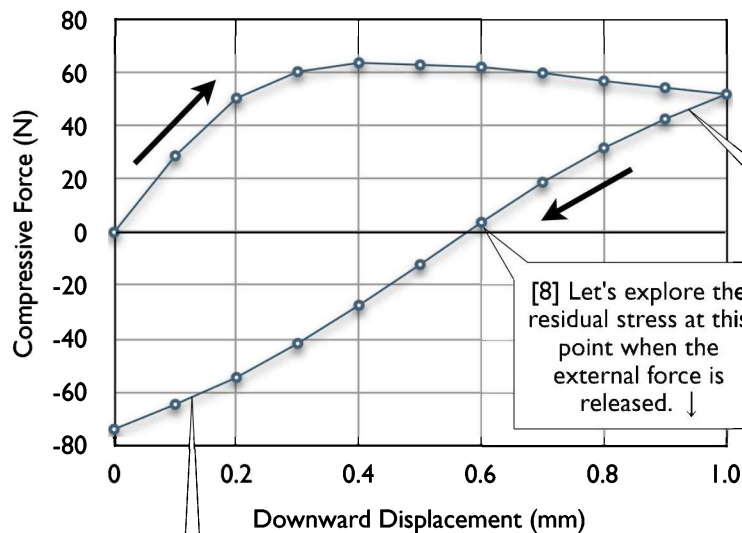
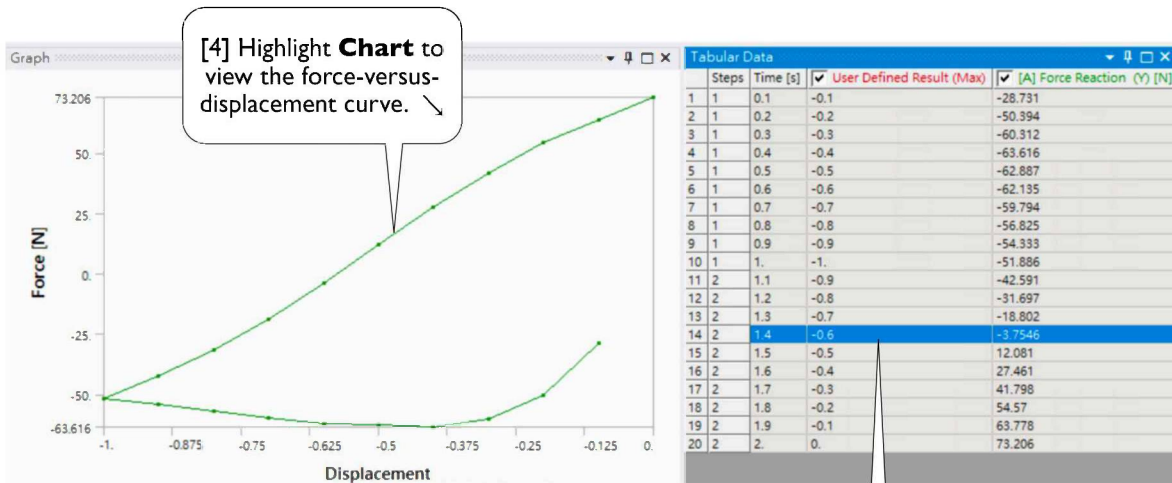
14.2.7 Results of the Nonlinear Material Simulation



[1] Highlight **Equivalent Stress**. A majority of the surface area is plasticized (stress larger than 250 MPa) at the end of the first load step. ←



[3] Highlight **Normal Stress** to view the hoop stress. Note that the top surface is dominated by compression (blue color), while the bottom surface is dominated by tension (red color). ↵



[7] The curve below zero force has little practical usage. It is the force required to pull the spring back to its original position. ↗

[9] Highlight **Equivalent Stress**. In the **Tabular Data**, click here (when the time is 1.4 s) and right-click-select **Retrieve This Result**. ↶

Tabular Data						
	Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]		
1	0.1	16.998	237.3	80.968		
2	0.2	25.347	263.72	143.23		
3	0.3	24.348	275.1	172.6		
4	0.4	18.371	278.15	187.95		
5	0.5	14.631	279.	199.21		
6	0.6	21.369	279.	208.03		
7	0.7	38.892	279.	215.96		
8	0.8	43.594	279.	223.85		
9	0.9	26.284	279.	230.66		
10	1.	10.201	279.	236.89		
11	1.1	24.043	213.02	169.62		
12	1.2	22.99	176.54	104.36		
13	1.3	3.8364	186.88	63.842		
14	1.4	5.8852	279.	78.716		
15	1.5	21.08	279.	119.97		
16	1.6	30.09	279.	161.14		
17	1.7			182.84		
18	1.8			192.22		
19	1.9			197.31		
20	2.			200.27		

Copy Cell

Retrieve This Result

Create Result

Export

Retrieve This Result

Select All

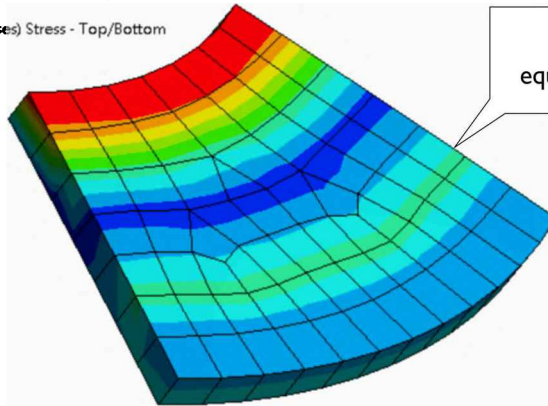
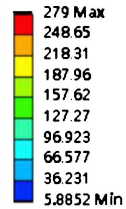
B: Belleville Washer (Nonlinear Material)

Equivalent Stress

Type: Equivalent (von-Mises) Stress - Top/Bottom

Unit: MPa

Time: 1.4



[10] Residual equivalent stress. ↓



[11] Highlight **Normal Stress**. In **Tabular Data**, click here (when the time is 1.4 s), and right-click-select **Retrieve This Result**. ↓

Time [s]	Minimum [MPa]	Maximum [MPa]	Average [MPa]
1 0.1	-240.2	136.19	10.268
2 0.2	-271.42	251.02	28.234
3 0.3	-281.42	261.8	30.492
4 0.4	-284.61	268.8	28.644
5 0.5	-285.43	272.54	27.24
6 0.6	-288.63	276.15	25.817
7 0.7	-290.49	277.55	24.946
8 0.8	-290.79	278.27	24.071
9 0.9	-290.8	279.09	23.338
10 1	-291.12	279.85	21.894
11 1.1	-222.82	202.29	17.474
12 1.2	-177.95	122.8	12.265
13 1.3	-140.77	185.71	6.4749
14 1.4	-106.85	285.19	-3.1354
15 1.5	-191.16	291.58	-21.191
16 1.6	-277.85	291.18	-40.004
17 1		3.45	-43.952
18 1		4.48	-41.417
19 1		6.71	-38.159
20 2		7.95	-35.028

B: Belleville Washer (Nonlinear Material)

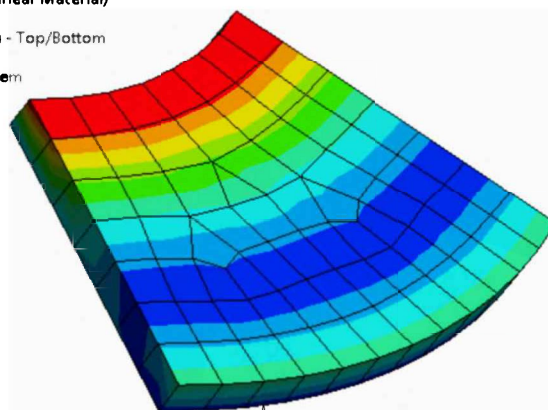
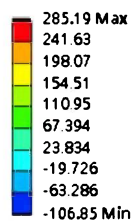
Normal Stress

Type: Normal Stress(Y Axis) - Top/Bottom

Unit: MPa

Cylindrical Coordinate System

Time: 1.4



[12] Residual hoop stress. Note that the top surface is dominated by tension (red color), while the bottom surface is dominated by compression (blue color; please compare this with the results in step [3], page 550). →

Wrap Up

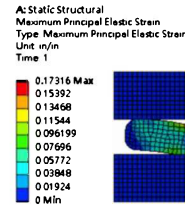
[13] Save the project and exit Workbench. #

Reference

1. Wikipedia>Belleville Washer.

Section 14.3

Planar Seal

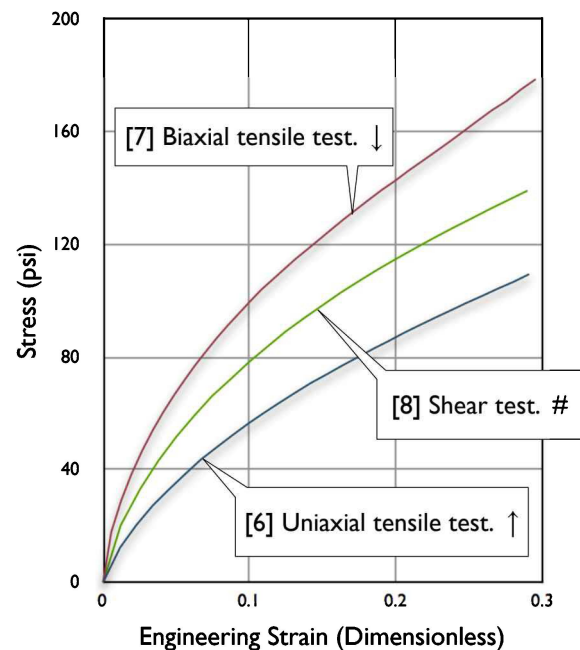


14.3.1 About the Planar Seal

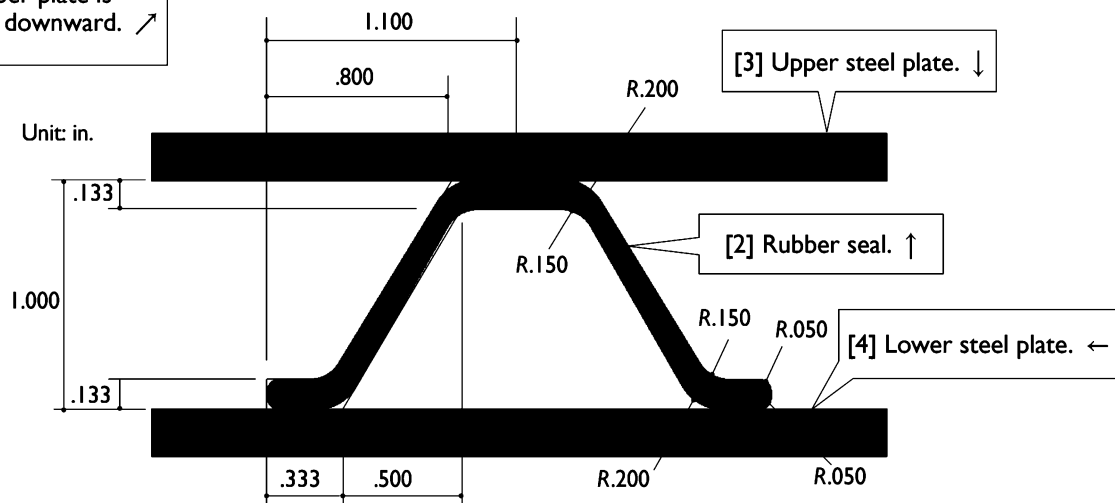
[1] The seal shown in [2-5] is used in the door of a refrigerator. The seal is a long strip of rubber, and we will model it as a plane strain problem (3.3.2, page 141). A series of material tests has been conducted, including a uniaxial tensile test, a biaxial tensile test, and a shear test [6-8].

The strain range of the original test data covers much more than the data shown in [6-8]. However, a preliminary study of the problem shows that the maximum strain does not exceed 0.3. Therefore, we decided to use the portion of data up to a strain of 0.3. The data fitting will be better if we use only the relevant data. A series of trials of data fitting shows that, for these material testing data, the two-parameter Mooney-Rivlin hyperelastic model fits the data better than other models. We decide to use two-parameter Mooney-Rivlin model.

The unit system used in this section is **in-lbm-lbf-s**.

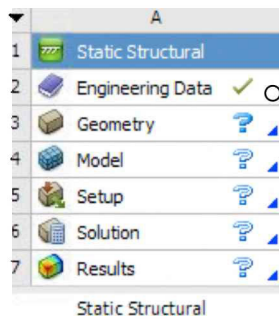


[5] The upper plate is displaced 0.85" downward. ↘

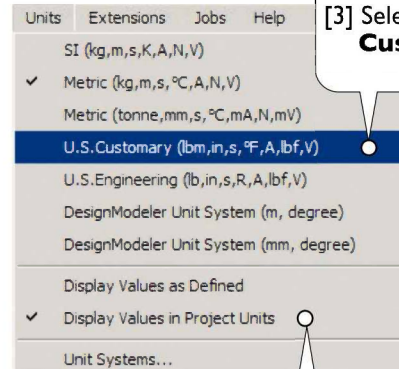


14.3.2 Prepare Material Properties for the Rubber

[1] Launch Workbench. Create a **Static Structural** analysis system. Save the project as **Seal**. ✓



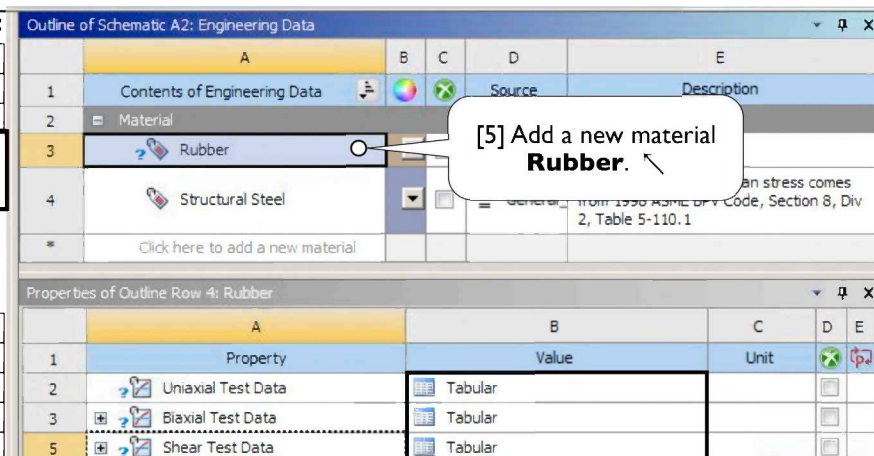
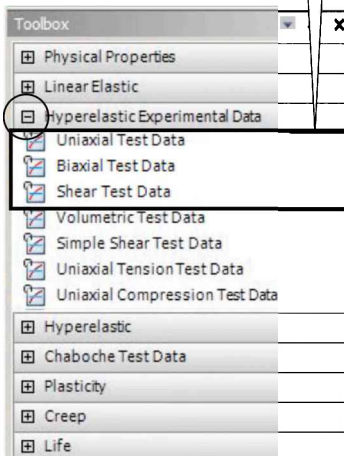
[2] Double-click to start up **Engineering Data**. →



[3] Select **Units/U.S. Customary**. ↓

[4] Make sure **Display Values in Project Units** is selected. ↓

[6] Expand **Hyperelastic Experimental Data** and double-click **Uniaxial Test Data**, **Biaxial Test Data**, and **Shear Test Data**. ↘



[5] Add a new material **Rubber**. ↘

Type or Copy Test Data?

[8] On the next page, we will show you how to input the test data [9-17]. We purposely simplify the data so that you should be able to type the data manually in a few minutes. However, if you really hate to type these data manually, you may import them from the following three files: **UniaxialTestData.csv**, **BiaxialTestData.csv**, and **ShearTestData.csv**. They are **comma-separated values** (CSV) files and can be downloaded from the SDC Publications website or the author's webpage. See **Preface** and the inside front cover for details. ↙

[7] We will input test data here. ←

Properties of Outline Row 4: Rubber

	A	B	C	D	E
1	Property	Value	Unit		
2	Uniaxial Test Data	Tabular			
3	Biaxial Test Data	Tabular			
5	Shear Test Data	Tabular			

[9] Click **Tabular** of **Uniaxial Test Data**. ✓

[12] Click **Tabular** of **Biaxial Test Data**. ↓

[10] Type data (or import the data from **UniaxialTestData.csv**, as demonstrated in [19-24], next page) like this. Of course, you may copy these data from the file and then paste here. ↓

	B	C
1	Strain (in/in)	Stress (psi)
2	0	0
3	0.0116	12.34
4	0.0227	20.35
5	0.0339	27.28
6	0.0451	33.06
7	0.0557	38.29
8	0.0664	43.45
9	0.0773	47.86
10	0.088	52.12
11	0.0989	56.3
12	0.1095	60.06
13	0.1203	63.74
14	0.131	67.28
15	0.1415	70.67
16	0.1524	73.77
17	0.1628	76.84
18	0.1737	79.87
19	0.184	82.8
20	0.1947	85.61
21	0.2051	88.5
22	0.2154	91.13
23	0.2259	93.77
24	0.2366	96.37
25	0.2475	99.07
26	0.2584	101.6
27	0.2687	104.09
28	0.2797	106.58
29	0.2903	109.29

	B	C
1	Strain (in/in)	Stress (psi)
2	0	0
3	0.0056	17.94
4	0.0124	29.12
5	0.0192	38.22
6	0.0264	46.45
7	0.0337	53.81
8	0.041	60.37
9	0.0481	66.18
10	0.0555	71.76
11	0.0628	77.06
12	0.0702	81.97
13	0.0775	86.67
14	0.0851	91.21
15	0.0924	95.29
16	0.1	99.56
17	0.1078	103.8
18	0.1154	107.33
19	0.1231	110.94
20	0.1312	114.72
21	0.1394	118.19
22	0.1475	121.68
23	0.1558	125.28
24	0.1642	128.86
25	0.1726	132.26
26	0.1813	135.78
27	0.19	139.2
28	0.1992	142.49
29	0.2083	146.07
30	0.2174	149.37
31	0.2266	152.75
32	0.2361	156.28
33	0.2458	160.03
34	0.2551	163.66
35	0.2649	167.41
36	0.2748	170.67
37	0.2847	174.77
38	0.2948	178.44

[15] Click **Tabular** of **Shear Test Data**. ↓

[13] Type or copy data (or import the data from **BiaxialTestData.csv**) like this. ↓

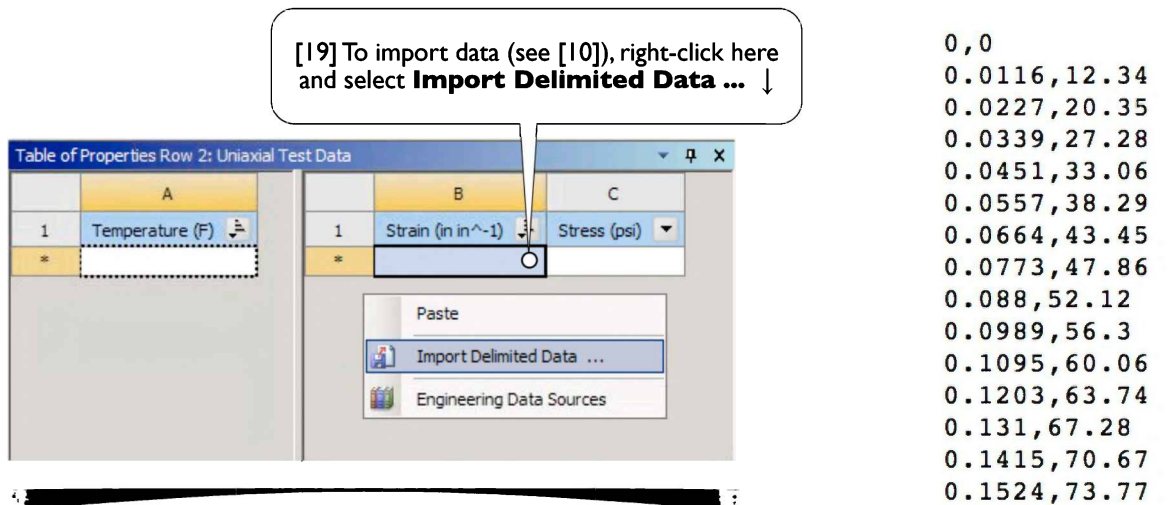
[16] Type or copy data (or import the data from **ShearTestData.csv**) like this. ↓

	B	C
1	Strain (in/in)	Stress (psi)
2	0	0
3	0.012	20.22
4	0.0252	33.06
5	0.0376	43.07
6	0.0499	51.64
7	0.0624	59.27
8	0.0744	66.15
9	0.0874	72.3
10	0.0999	78.36
11	0.1124	83.77
12	0.1243	88.93
13	0.1371	93.76
14	0.1498	98.36
15	0.1619	102.75
16	0.1744	106.86
17	0.187	110.9
18	0.1998	114.76
19	0.2126	118.48
20	0.2252	122.09
21	0.238	125.61
22	0.2511	129.05
23	0.2633	132.32
24	0.2763	135.64
25	0.2892	138.91

	A
1	Temperature (F)
2	70

[11, 14, 17] In **Table of Properties**, type 70 (degree F) for **Temperature**. ↑ ↑ →

[18] Go to [25], page 557. ↩



```

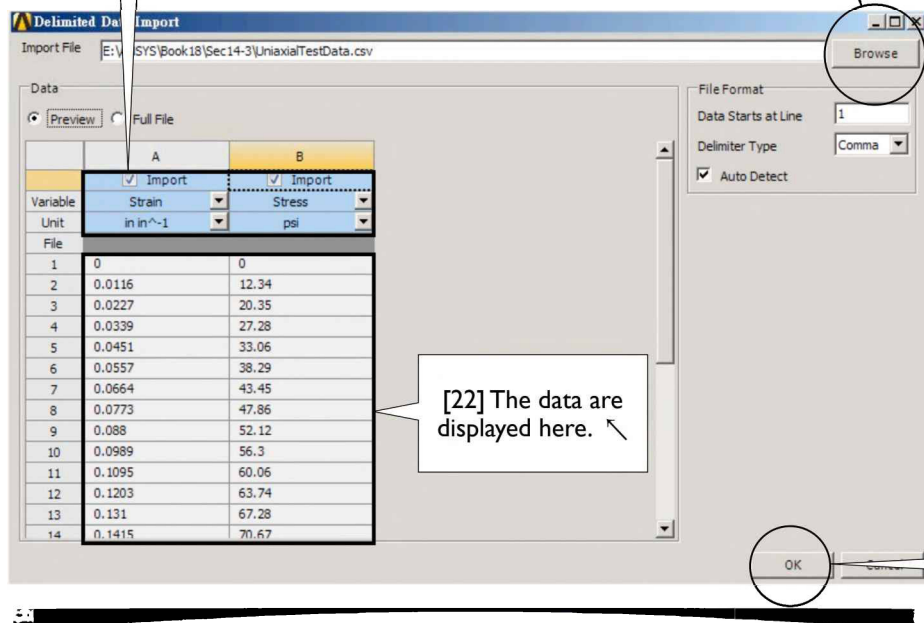
0,0
0.0116,12.34
0.0227,20.35
0.0339,27.28
0.0451,33.06
0.0557,38.29
0.0664,43.45
0.0773,47.86
0.088,52.12
0.0989,56.3
0.1095,60.06
0.1203,63.74
0.131,67.28
0.1415,70.67
0.1524,73.77
0.1628,76.84
0.1737,79.87
0.184,82.8
0.1947,85.61
0.2051,88.5
0.2154,91.13
0.2259,93.77
0.2366,96.37
0.2475,99.07
0.2584,101.6
0.2687,104.09
0.2797,106.58
0.2903,109.29

```

[21] The file **UniaxialTestData.csv** is a text file, containing **comma-separated values (csv)** like this. ↓

[20] Click **Browse** and open the file **UniaxialTestData.csv**. ↑

[23] Set up the variables like this. ↘



[25] Expand **Hyperelastic** and double-click **Mooney-Rivlin 2 Parameter**. ✓

[26] The material model is included here. Expand it. ↓

[27] Right-click **Curve Fitting** and select **Solve Curve Fit**. ✓

[28] These are the calculated material parameters. ↓

[29] Dotted data are test data, and the solid curves are fitted curves. ↓

[30] Right-click **Curve Fitting** again and select **Copy Calculated Values to Property**. ↓

[31] Now, the material parameters are input here. You could have directly input these material parameters here without steps [6-24, 27-30] (steps [25-26] are required). ↓

[32] Return to **Project Schematic**. #

Table of Properties Row 17: Mooney-Rivlin 2 Parameter

Coefficient Name	Calculated Value	Calculated Unit
Incompressibility Parameter D1	0	psi ⁻¹
Material Constant C01	-4.8079	psi
Material Constant C10	103.17	psi
Residual	3.4402	

Chart of Properties Row 17: Mooney-Rivlin 2 Parameter

Stress [psi] vs Strain [in in⁻¹]

The chart displays experimental data points (dotted lines) and fitted curves (solid lines) for Uniaxial Test Data, Shear Test Data, and Biaxial Test Data.

Properties of Chart : Uniaxial Test Data Rubber

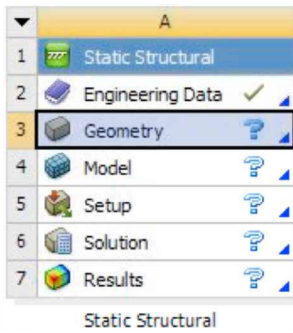
Property	Value	Unit
Uniaxial Test Data	Tabular	
Biaxial Test Data	Tabular	
Shear Test Data	Tabular	
Mooney-Rivlin 2 Parameter		
Material Constant C10	103.17	psi
Material Constant C01	-4.8079	psi
Incompressibility Parameter D1	0	psi ⁻¹
Curve Fitting	Fit Type: Mooney-Rivlin 2 Parameter	
Error Norm for Fit	Normalized Error	
Uniaxial Test Data	Tabular	
Biaxial Test Data	Tabular	
Shear Test Data	Tabular	
Volumetric Test Data	Add this experimental data, to include it in the curve fitting.	

Properties of Chart : Uniaxial Test Data Rubber

Property	Value	Unit
Uniaxial Test Data	Tabular	
Biaxial Test Data	Tabular	
Shear Test Data	Tabular	
Mooney-Rivlin 2 Parameter		
Material Constant C10	103.17	psi
Material Constant C01	-4.8079	psi
Incompressibility Parameter D1	0	psi ⁻¹
Curve Fitting	Fit Type: Mooney-Rivlin 2 Parameter	
Error Norm for Fit	Normalized Error	
Uniaxial Test Data	Tabular	
Biaxial Test Data	Tabular	
Shear Test Data	Tabular	
Volumetric Test Data	Add this experimental data, to include it in the curve fitting.	

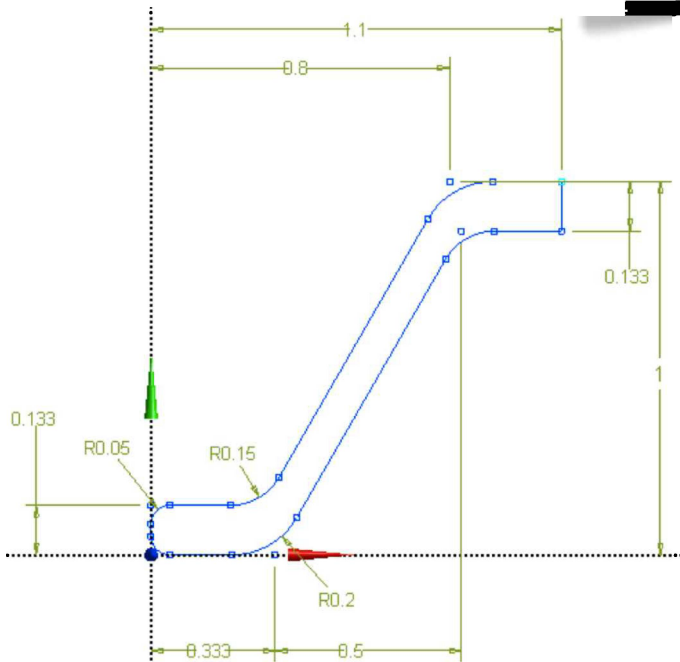
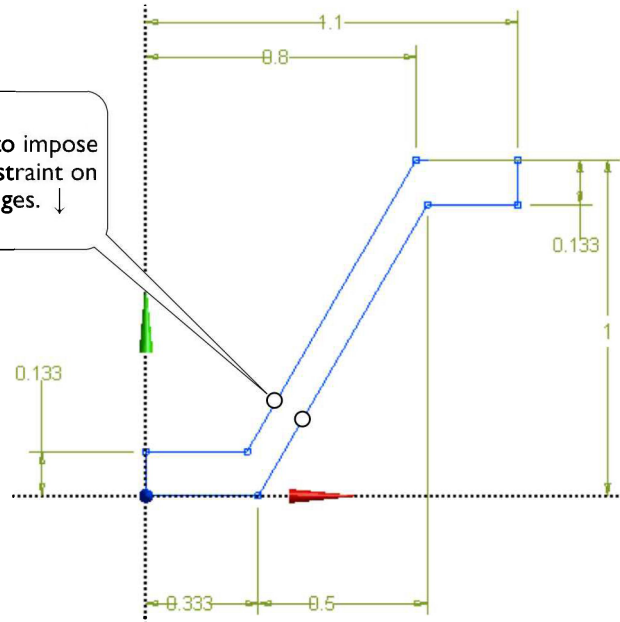
Project Schematic

14.3.3 Create 2D Geometry

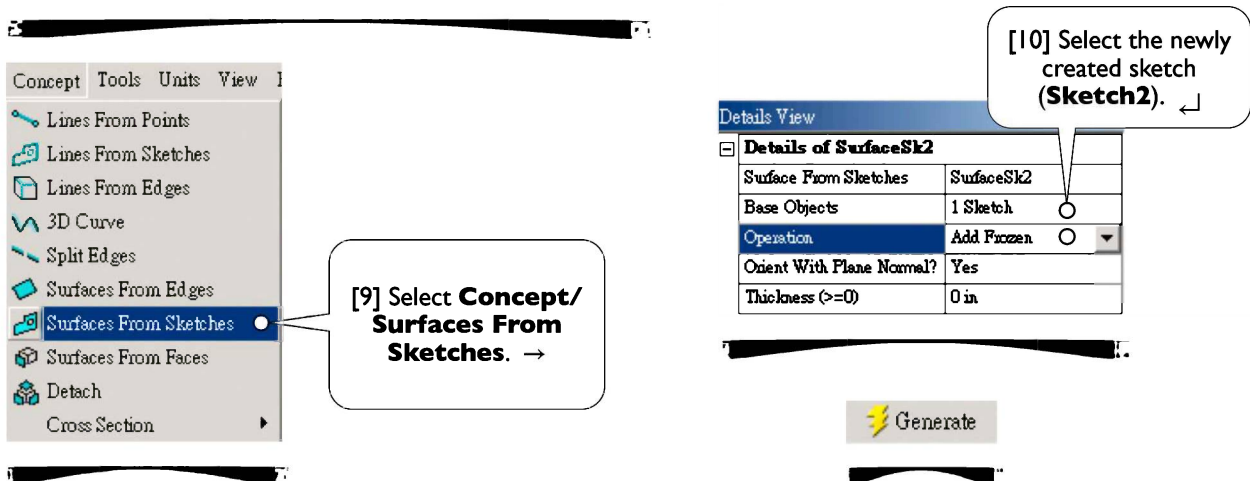
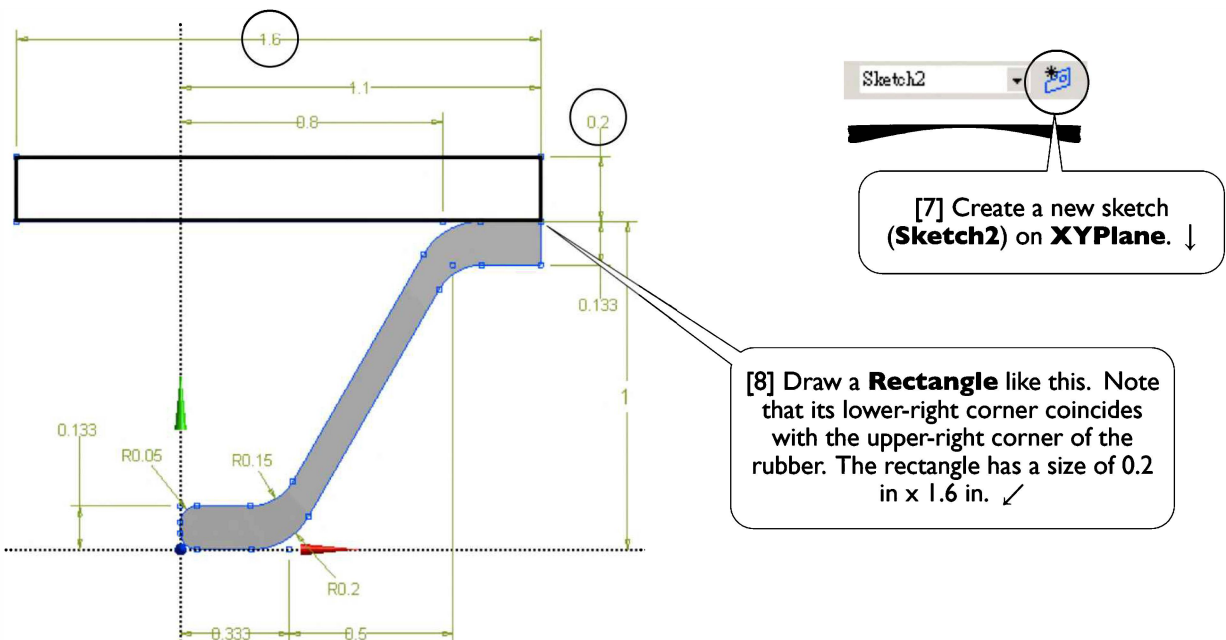
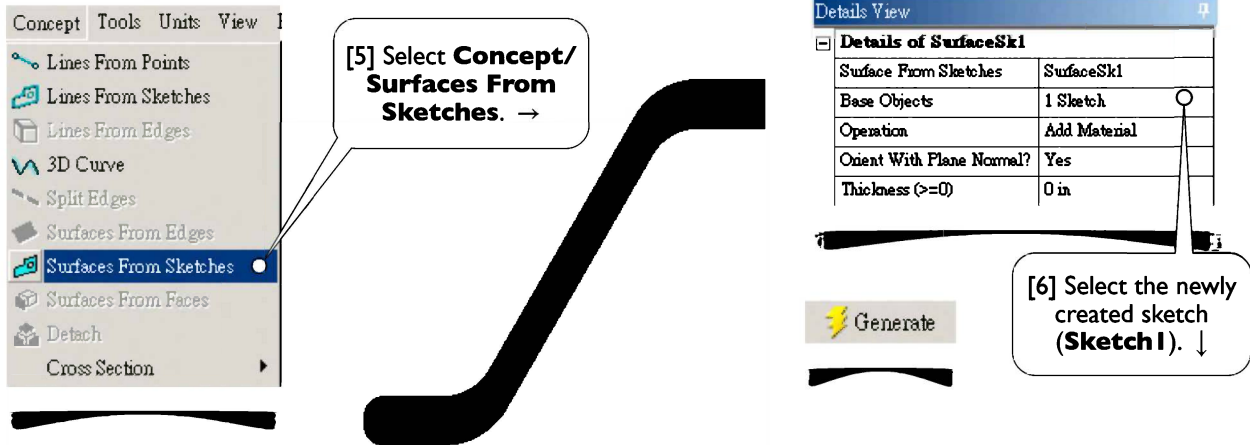


[1] Start up DesignModeler. ✓

[2] Select **Inch** as the length unit. Create a sketch on **XYPlane** like this. ↑



[4] Add fillets and specify their radii. (For the radii of the fillets, please also see 14.3.1, page 553.) ↵



[11] Create a new sketch (Sketch3) on XYPlane. ↓

[12] Draw a rectangle of the same size as the upper plate. Note that its upper edge coincides with the horizontal axis. ↓

[13] Select **Concept/Surfaces From Sketches. ↓**

[14] Select the newly created sketch (Sketch3). ↓

[15] Rename the three bodies. Close DesignModeler. #

Concept Tools Units View

- Lines From Points
- Lines From Sketches
- Lines From Edges
- 3D Curve
- Split Edges
- Surfaces From Edges
- Surfaces From Sketches**
- Surfaces From Faces
- Detach
- Cross Section

Details View

Details of SurfaceSk3	
Surface From Sketches	SurfaceSk3
Base Objects	1 Sketch
Operation	Add Frozen
Orient With Plane Normal?	Yes
Thickness (>=0)	0 in

A: Static Structural

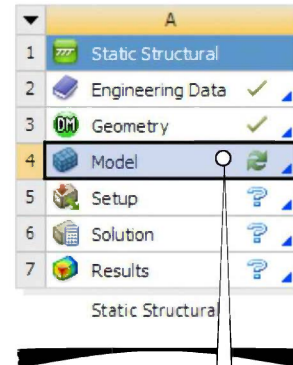
- XYPlane
- Sketch1
- Sketch2
- Sketch3
- ZXPlane
- YZPlane
- SurfaceSk1
- SurfaceSk2
- SurfaceSk3
- 3 Parts, 3 Bodies
 - Rubber
 - Upper Plate
 - Lower Plate

14.3.4 Set Up for Simulation

Properties of Schematic A3: Geometry

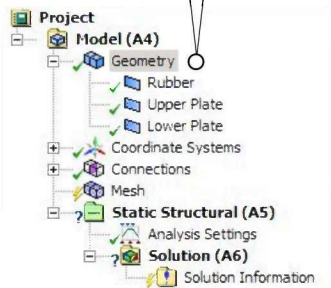
	A	B
1	Property	Value
2	General	
3	Component ID	Geometry
4	Directory Name	SYS
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	Geometry Source	
10	Geometry File Name	
11	Advanced Geometry Options	
12	Analysis Type	2D
13	Compare Parts On Update	No

[1] Select **2D** for **Analysis Type** (3.1.4[4-6], page 112). ↘



[2] Start up **Mechanical**. Select the **in-lbm-lbf-s** unit system. ←

[3] Highlight **Geometry**. ↓

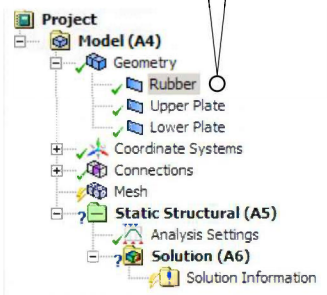


Details of "Geometry"

Definition	
Source	DesignModeler
Type	Meters
Length Unit	Meters
Element Control	Program Controlled
2D Behavior	Plane Strain
Display Style	Body Color
Bounding Box	
Properties	
Statistics	
Basic Geometry Options	
Advanced Geometry Options	

[4] Select **Plane Strain** (3.3.2, page 141) for **2D Behavior**. ↗

[5] Highlight **Rubber**. ↓

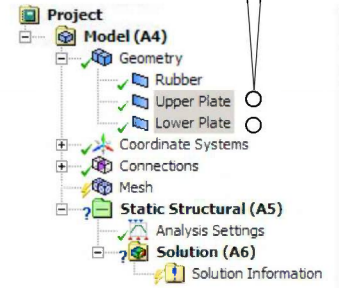


Details of "Rubber"

Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Behavior	None
Material	
Assignment	Rubber
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	
Statistics	
CAD Attributes	
DMSheetThickness	0

[6] Select **Rubber** for **Assignment**. ↗

[7] Select **Upper Plate** and then control-select **Lower Plate**. ↓



Details of "Multiple Selection"

Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Behavior	None
Material	
Assignment	Structural Steel
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	
Statistics	

[8] Make sure the material is **Structural Steel**. ←

Detail of "Frictionless - Rubber To Upper Plate"	
Scope	
Scoping Method	Geometry Selection
Contact	7 Edges
Target	1 Edge
Contact Bodies	Rubber
Target Bodies	Upper Plate
Shell Thickness Effect	No
Definition	
Type	Frictionless
Scope Mode	Manual
Behavior	Asymmetric
Tie Contact	Program Controlled
Suppressed	No
Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Stabilization Damping Factor	0.
Pinball Region	Program Controlled
Time Step Controls	None
Geometric Modification	
Interface Treatment	Add Offset, No Ramping
Offset	0. mm
Contact Geometry Correction	None
Target Geometry Correction	None

[9] Highlight **Connections/Contacts/Contact Region**, and reselect the contact edges, a total of 7 edges selected, see [10-11]. →

[11] End of the contact edges, including the vertical edge. ↓

[10] Start of the contact edges. ←

[12] Highlight **Connections/Contacts/Contact Region 2**, reselect the contact edges, a total of 6 edges selected, see [13-14]. →

[14] End of the contact edges, not including the vertical edge. ✓

[13] Start of the contact edges. ✓

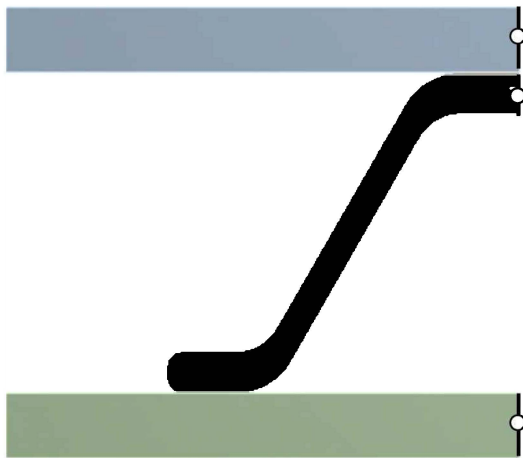
Detail of "Frictionless - Rubber To Lower Plate"	
Scope	
Scoping Method	Geometry Selection
Contact	6 Edges
Target	1 Edge
Contact Bodies	Rubber
Target Bodies	Lower Plate
Shell Thickness Effect	No
Definition	
Type	Frictionless
Scope Mode	Manual
Behavior	Asymmetric
Tie Contact	Program Controlled
Suppressed	No
Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Stabilization Damping Factor	0.
Pinball Region	Program Controlled
Time Step Controls	None
Geometric Modification	
Interface Treatment	Add Offset, No Ramping
Offset	0. mm
Contact Geometry Correction	None
Target Geometry Correction	None

Asymmetric Behavior

[15] When flat surfaces (or edges) are involved in a contact region, the flat surfaces are usually chosen as **Target**. For these cases, we may choose **Asymmetric** for **Behavior** to save computing time. →

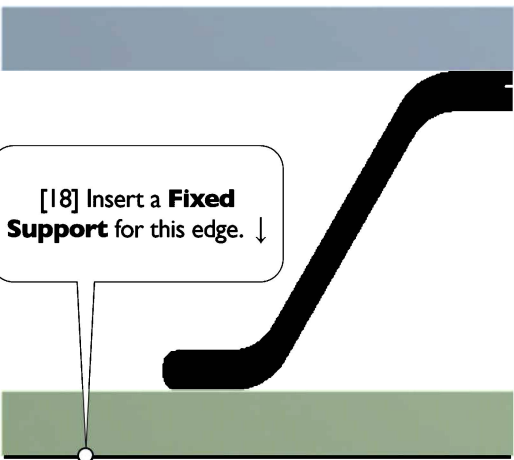
Detail of "Analysis Settings"	
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Restart Controls	
Nonlinear Controls	
Output Controls	
Analysis Data Management	

[16] Turn on **Large Deflection**. ←



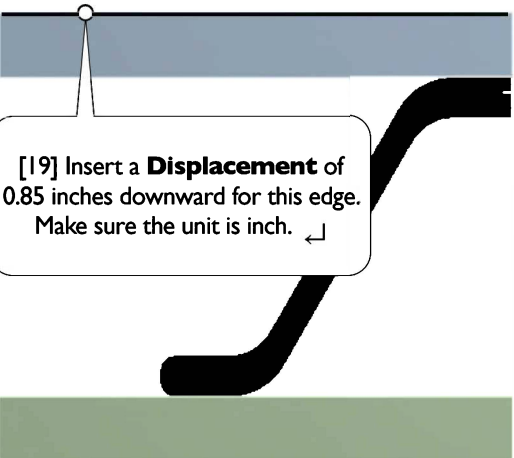
[17] Insert a **Frictionless Support** for these three edges (they are on the plane of symmetry). ✓

Details of "Frictionless Support"	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	3 Edges <input type="radio"/>
[-] Definition	
Type	Frictionless Support
Suppressed	No



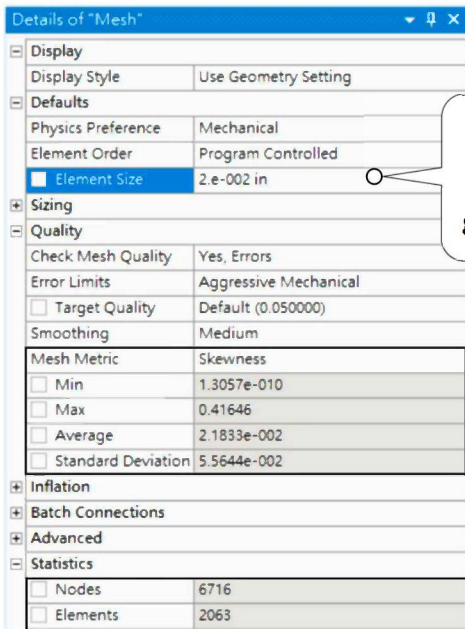
[18] Insert a **Fixed Support** for this edge. ↓

Details of "Fixed Support"	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge <input type="radio"/>
[-] Definition	
Type	Fixed Support
Suppressed	No

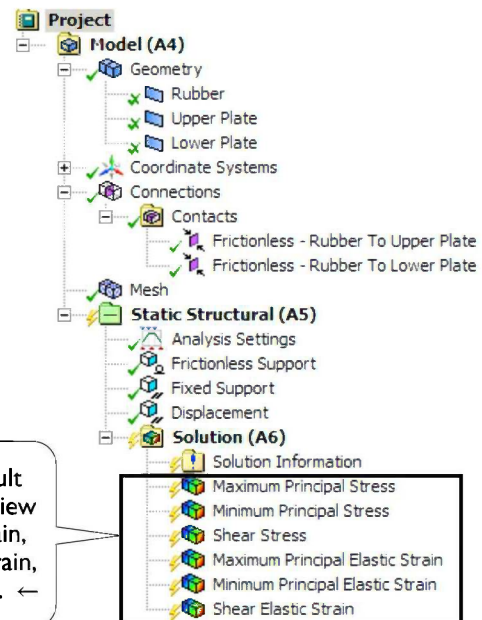
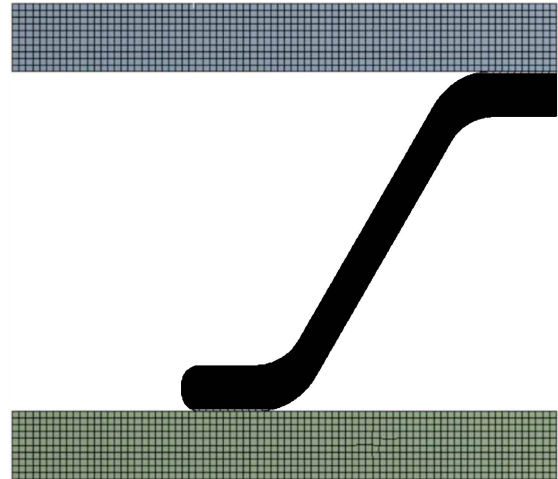


[19] Insert a **Displacement** of 0.85 inches downward for this edge. ↵

Details of "Displacement"	
[-] Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge <input type="radio"/>
[-] Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
<input checked="" type="checkbox"/> Y Component	-0.85 in. (ramped) <input type="radio"/>
Suppressed	No

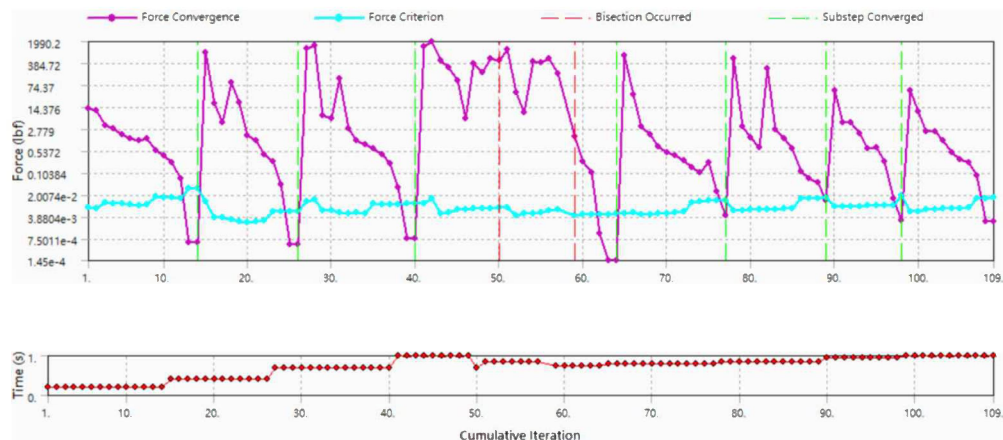


[20] Set the element size to 0.02 inches and generate mesh. ↓

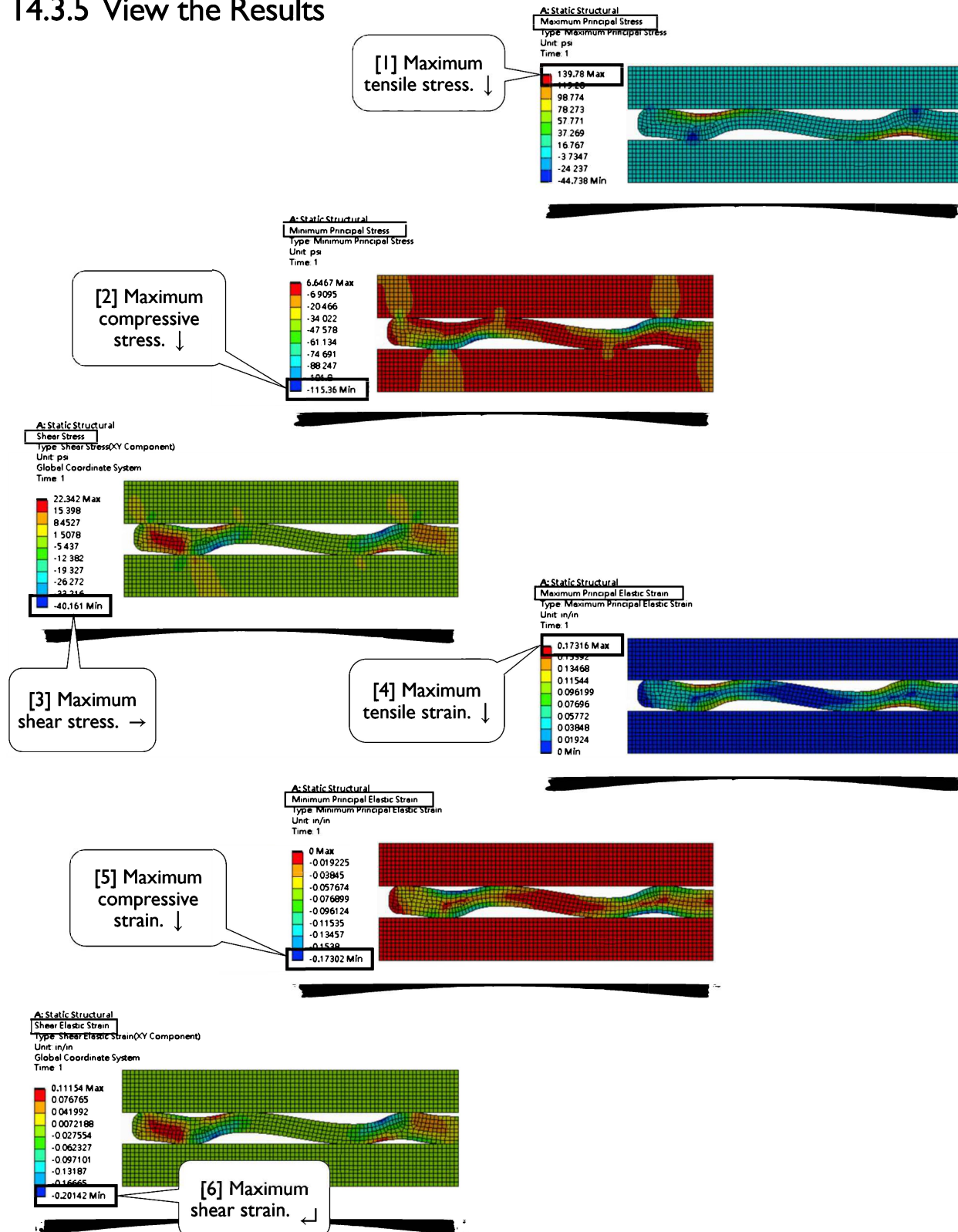


[22] Solve. #

[21] Insert these result objects. We want to view the tensile stress/strain, compressive stress/strain, and shear stress/strain. ←



14.3.5 View the Results



[7] Animate the results. ↓

[8] With **Solution** highlighted, insert a **Probe/Force Reaction**. ↗

[9] Select **Displacement**. ↓

[10] Solve. ✓

[11] A force history curve. →

[12] For a plane strain problem, Workbench assumes a unit depth (i.e., in this case, one inch). Therefore the force unit should be read **lbf/in** (rather than **lbf**). #

The 'Details of "Force Reaction"' panel shows the following settings:

- Definition**
 - Type: Force Reaction
 - Location Method: Boundary Condition
 - Boundary Condition: Displacement
 - Orientation: Global Coordinate System
 - Suppressed: No
- Options**
 - Result Selection: Y Axis
 - Display Time: End Time
- Results**
 - Maximum Value Over Time: Y Axis
 - Minimum Value Over Time: Y Axis
- Information**

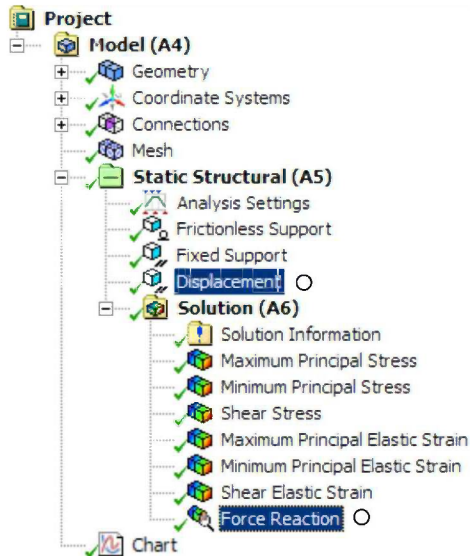
The 'Solve' button is highlighted with a lightning bolt icon.

The 'Force Reaction (Y) [lbf]' graph shows a force history curve over time. The y-axis is labeled [lbf] and ranges from -4.0808e-2 to -4.1679. The x-axis is labeled [s] and ranges from 0 to 1. The curve starts at approximately -4.0808e-2 at t=0, remains relatively flat until t=0.75, and then drops sharply to -4.1679 at t=1.

The 'Tabular Data' table shows the following data:

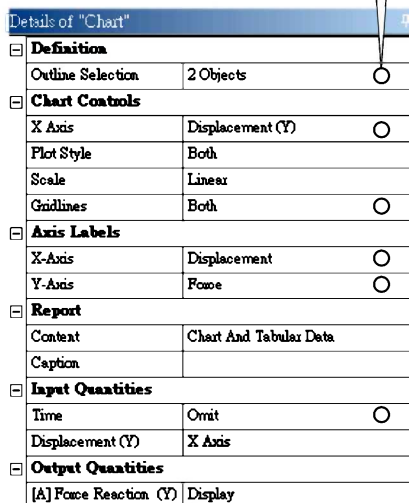
Time [s]	Force Reaction (Y) [lbf]
1 0.2	-4.0808e-002
2 0.4	-7.0668e-002
3 0.7	-0.11459
4 0.7525	-0.12804
5 0.805	-0.34942
6 0.8575	-0.71464
7 0.93625	-1.2486
8 1.	-4.1679

14.3.6 Create a Force-versus-Displacement Chart

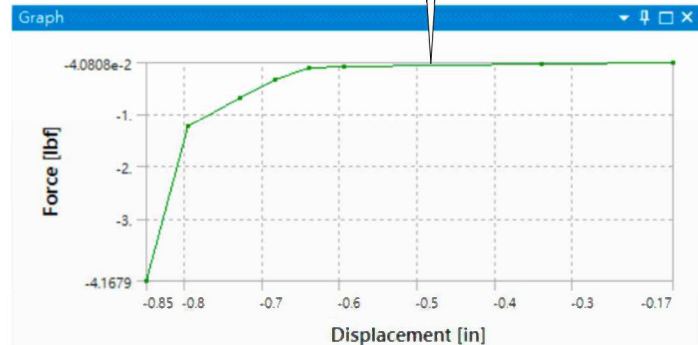


[1] Click **Chart**. ✓

[2] Click to bring up **Apply/Cancel** buttons and control-select **Static Structural/Displacement** and **Solution/Force Reaction**. →



[3] A force-versus-displacement curve. Note that the force unit should be read **lbf/in** (instead of **lbf**; also see 14.3.5[12], last page). ↓



[4] This curve is plotted using only 8 points since the model is solved with 8 substeps (see 14.3.5[12], last page). To obtain more data points for a more smooth curve, you may need to change the settings in **Analysis Settings**. ←

Wrap Up

[5] Save the project and exit Workbench. #

Section 14.4

Review

14.4.1 Keywords

Choose a letter for each keyword, from the list of descriptions

- | | |
|---------------------------------|------------------------------------|
| 1. () Elastic Material | 9. () Nonlinear Materials |
| 2. () Hardening Rules | 10. () Plastic Materials |
| 3. () Hyperelastic Materials | 11. () Rate-Dependent Materials |
| 4. () Kinematic Hardening | 12. () Subsequent Yield Surface |
| 5. () Initial Yield Point | 13. () Time-Dependent Materials |
| 6. () Initial Yield Surface | 14. () Von Mises Yield Surface |
| 7. () Isometric Hardening | 15. () Yield Surface |
| 8. () Linear Materials | |

Answers:

1. (B) 2. (M) 3. (G) 4. (N) 5. (H) 6. (K) 7. (O) 8. (A) 9. (E) 10. (F)
 11. (D) 12. (L) 13. (C) 14. (J) 15. (I)

List of Descriptions

(A) A material that has a linear stress-strain relation and can be described by Hooke's law. In this book, a linear material also implies elasticity, time-independent, and rate-independent.

(B) Materials in which the strain returns to its original state when the stress is removed.

(C) Also called viscous materials. If you apply a constant stress on the materials, the occurrence of strain is time-dependent. The strain typically increases with time and stabilizes to a constant value after a period of time. This behavior is called creeping. On the other hand, if you apply a constant strain on the materials, the occurrence of stress is also time-dependent. The stress typically decreases with time and stabilizes to a constant value after a period of time. This behavior is called stress-relaxation.

(D) Materials in which the stress-strain relation is such that the magnitude of stress depends on not only the magnitude of strain but also the rate of the strain, and vice versa.

- (E) In this book, all materials that are not described by Hooke's law are categorized as nonlinear materials.
- (F) Materials that exhibit plasticity behavior, which is characterized by strain that does not return to its original state after the stress is removed. Rather, residual strain, or plastic strain, remains.
- (G) Materials that remain elastic under a very large strain.
- (H) In plasticity, yield point may change. Stress at the elastic limit is called the initial yield point.
- (I) In a uniaxial test, yielding occurs at certain stress values; they are called yield points. In 3D cases, yielding occurs at certain stress states; these stress states form a "surface" (or hyper-surface) in the multiaxial stress space. The surface is called a yield surface. During stressing, the yield surface may change in size as well as location.
- (J) Different yield criterion results in different yield surface. If von Mises yield criterion is used, the yield surface is a cylindrical surface in the σ_1 - σ_2 - σ_3 space.
- (K) In plasticity, yield surface may change in size as well as location during stressing. The original yield surface before any yielding occurs (i.e., the behavior is still elastic) is called the initial yield surface.
- (L) In plasticity, during stressing, yield surface may change in size as well as location. All possible yield surfaces except the initial yield surface are called the subsequent yield surfaces.
- (M) Rules that describe how yield surface changes its size and location.
- (N) The yield surface changes only the location but not the size.
- (O) The yield surface changes only the size but not the location.

14.4.2 Additional Workbench Exercises

Model the Belleville Washer as a 2D Problem

In Section 14.2, we modeled the Belleville washer as a surface body and meshed it with shell elements. It is possible to model it as a 2D problem and mesh it with 2D axisymmetric solid elements. Do it and draw the pros and cons for both modeling methods.

Pneumatic Finger

Replace the material of the pneumatic finger, simulated in Section 9.1, with the rubber used in the Section 14.3. Redo the simulation for the pneumatic finger.

Chapter 15

Explicit Dynamics

Many transient dynamic simulations require extremely small integration time steps, for example, high-speed impact, drop test, or highly nonlinear problems. In such cases, the time steps may be as small as a few nanoseconds, and the use of **Transient Structural** analysis system becomes impractical, since the run time would be too enormous. The integration method used in **Transient Structural** analysis system is an *implicit method*.

Explicit Dynamics analysis system also deals with transient structural dynamics; however, it uses an *explicit integration method*. The *explicit method* is very efficient for each time step. It thus allows a large number of time steps to calculate within an acceptable time. A characteristic of explicit methods is that the integration time steps must be very small (e.g., microseconds or nanoseconds) in order to achieve stable solutions. If the dynamic behavior must be observed within a long duration (say, several seconds), then many millions of time steps are required to complete the simulation. In these situations, high-performance computing facilities are usually used to facilitate the computations.

Purpose of This Chapter

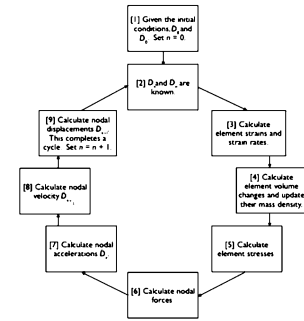
This chapter provides the basics of explicit dynamics so that the students have enough background knowledge to perform some simple explicit dynamics simulations. A high-speed impact simulation and a drop test simulation are used to demonstrate the applications of **Explicit Dynamics**.

About Each Section

Section 15.1 provides the basics of explicit dynamics. Section 15.2 presents a high-speed impact example and Section 15.3 presents a drop test simulation. Both examples are simple yet instructional. We use default settings as much as possible, so that the students can appreciate the built-in capabilities of **Explicit Dynamics**.

Section 15.1

Basics of Explicit Dynamics



15.1.1 Implicit Integration Methods

[1] As mentioned, transient dynamic simulations involve solving the equation,

$$[M]\{\ddot{D}\} + [C]\{\dot{D}\} + [K]\{D\} = \{F\} \quad \text{Copy of Eq. 12.1.4(1), page 426}$$

Consider a typical time step at t_n . Let D_n, \dot{D}_n , and \ddot{D}_n be the displacement, velocity, and acceleration at t_n , and D_{n+1}, \dot{D}_{n+1} , and \ddot{D}_{n+1} at t_{n+1} . Also, let $\Delta t = t_{n+1} - t_n$. We temporarily assume that the acceleration is linear over the time step (i.e., $\ddot{D}_n = \ddot{D}_{n+1} = 0$), then, by Taylor series expansions at t_n ,

$$\dot{D}_{n+1} = \dot{D}_n + \Delta t \ddot{D}_n + \frac{\Delta t^2}{2} \ddot{\ddot{D}}_n \quad (1)$$

$$D_{n+1} = D_n + \Delta t \dot{D}_n + \frac{\Delta t^2}{2} \ddot{D}_n + \frac{\Delta t^3}{6} \ddot{\ddot{D}}_n \quad (2)$$

The quantity $\ddot{\ddot{D}}_n$ can be approximated by

$$\ddot{\ddot{D}}_n = \frac{\ddot{D}_{n+1} - \ddot{D}_n}{\Delta t} \quad (3)$$

Substitution of Eq. (3) into Eqs. (1) and (2) respectively yields

$$\dot{D}_{n+1} = \dot{D}_n + \frac{\Delta t}{2} (\ddot{D}_{n+1} + \ddot{D}_n) \quad (4)$$

$$D_{n+1} = D_n + \Delta t \dot{D}_n + \Delta t^2 \left(\frac{1}{6} \ddot{D}_{n+1} + \frac{1}{3} \ddot{D}_n \right) \quad (5)$$

Eqs. (4) and (5) can be regarded as a special case of Newmark methods,

$$\dot{D}_{n+1} = \dot{D}_n + \Delta t [\gamma \ddot{D}_{n+1} + (1 - \gamma) \ddot{D}_n] \quad (6)$$

$$D_{n+1} = D_n + \Delta t \dot{D}_n + \frac{1}{2} \Delta t^2 [2\beta \ddot{D}_{n+1} + (1 - 2\beta) \ddot{D}_n] \quad (7)$$

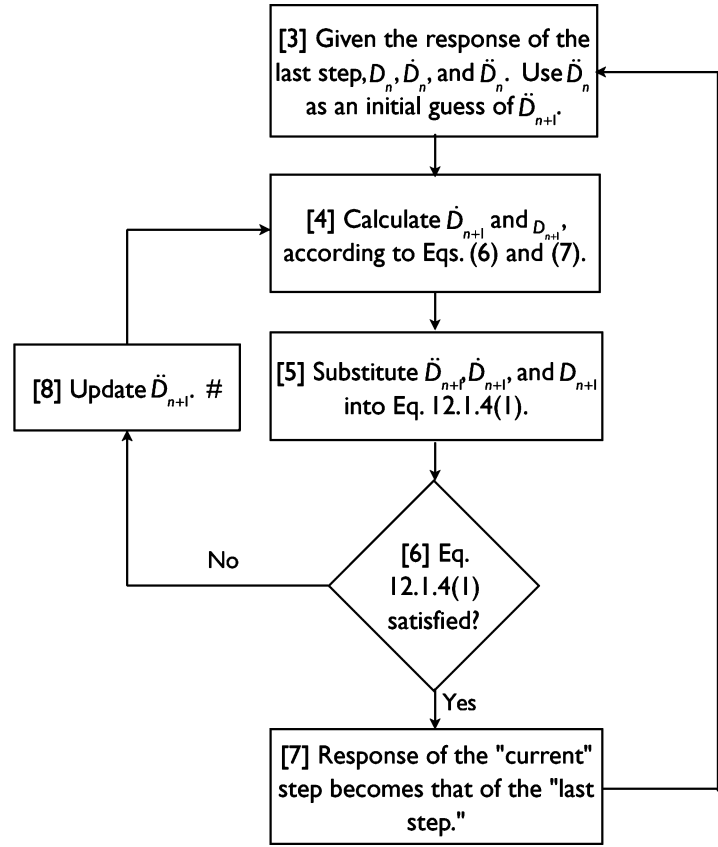
If you substitute $\gamma = 1/2$ and $\beta = 1/6$ into Eqs. (6) and (7) respectively, you will come up with Eqs. (4) and (5).

Eqs. (6) and (7) are used in **Transient Structural** analysis system. The parameters γ and β are chosen to control characteristics of the algorithm such as accuracy, numerical stability, etc. It is called an *implicit method* because the calculation of \dot{D}_{n+1} and D_{n+1} requires knowledge of \ddot{D}_{n+1} . That is, the response at the current time step depends on not only the historical information but also the current information; therefore, solving Eqs. (6) and (7) involves iterative process. ↵

[2] Calculation of the response at time t_{n+1} is depicted in [3-8]. In the beginning [3], the displacement D_n , velocity \dot{D}_n , and acceleration \ddot{D}_n of the last step are already known (For $n = 0$, we may assume $\ddot{D}_0 = 0$). Since \ddot{D}_{n+1} is needed in Eqs. (6) and (7), we use \ddot{D}_n as an initial guess of \ddot{D}_{n+1} . Knowing D_n , \dot{D}_n , and \ddot{D}_{n+1} , the quantities \dot{D}_{n+1} and D_{n+1} can be calculated according to Eqs. (6) and (7) [4]. The next step [5] is to substitute \ddot{D}_{n+1} , \dot{D}_{n+1} , and D_{n+1} into Eq. 12.1.4(1). If Eq. 12.1.4(1) is satisfied [6], then the calculation of the response at time t_{n+1} is complete [7]; otherwise, \ddot{D}_{n+1} is updated and another iteration is initiated [8]. Update of \ddot{D}_{n+1} [8] is similar to the Newton-Raphson method described in 13.1.4, page 472.

With implicit methods, integration time step is typically about milliseconds; a typical simulation time is about 0.1 to 10 seconds, which requires hundreds to ten-thousands of integration time steps.

Implicit methods can be used for most transient structural simulations. However, for highly nonlinear problems, it often fails due to convergence issues; for high-speed impact problems, the integration time is so small that the computing time becomes intolerable. In such cases, explicit methods are more applicable. ↗



15.1.2 Explicit Integration Methods

[1] The explicit method used in **Explicit Dynamics** analysis system is based on half-step central differences

$$\ddot{D}_n = \frac{\dot{D}_{n+\frac{1}{2}} - \dot{D}_{n-\frac{1}{2}}}{\Delta t}, \text{ or } \dot{D}_{n+\frac{1}{2}} = \dot{D}_{n-\frac{1}{2}} + \ddot{D}_n \Delta t \quad (1)$$

$$\dot{D}_{n+\frac{1}{2}} = \frac{D_{n+1} - D_n}{\Delta t}, \text{ or } D_{n+1} = D_n + \dot{D}_{n+\frac{1}{2}} \Delta t \quad (2)$$

Eqs. (1) and (2) are called *explicit methods* because the calculation of $\dot{D}_{n+\frac{1}{2}}$ and D_{n+1} requires knowledge of historical information only. That is, the response at the current time can be calculated explicitly; no iterations within a time step are needed. Therefore, it is very efficient to complete a time step, also called a *cycle*. One of the distinct characteristics of the explicit method is that its integration time step needs to be very small to achieve a stable solution. ↘

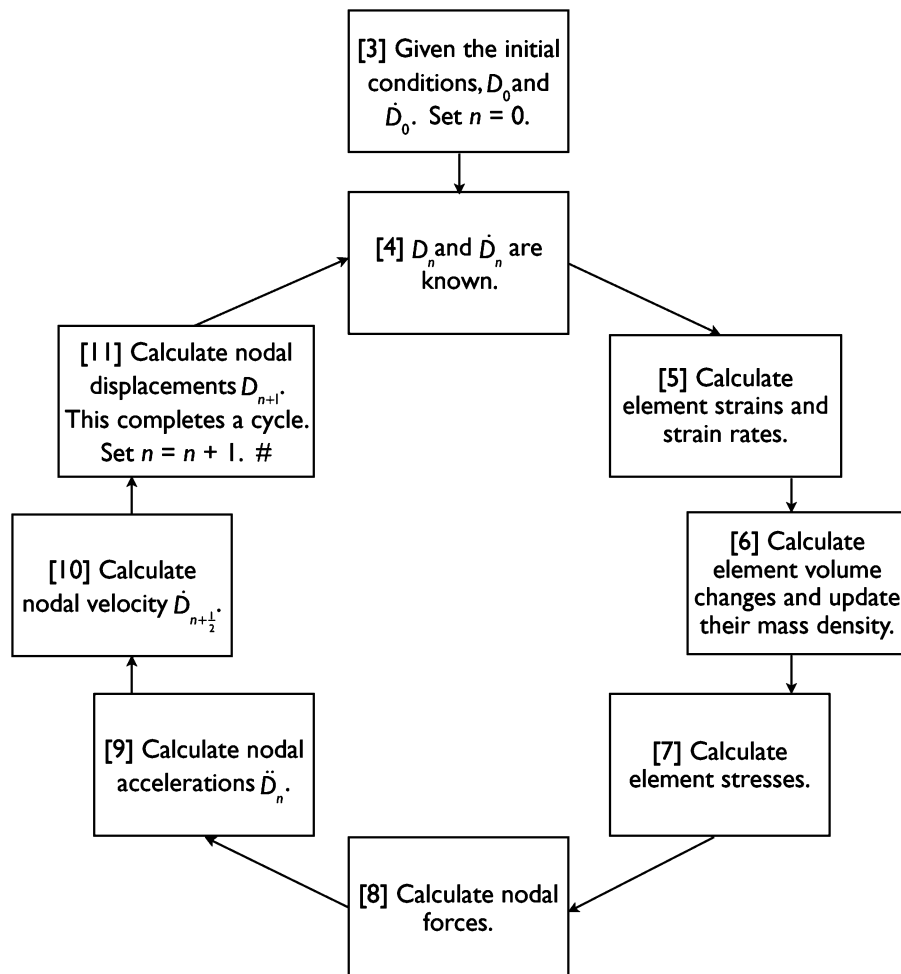
[2] The procedure used in the **Explicit Dynamics** analysis system is illustrated in [3-11]. In the beginning of a cycle [3-4], the displacement D_n and velocity \dot{D}_n of the last cycle are already known. With this information, we can calculate the strain and strain rate for each element [5], using the relations such as Eqs. 1.3.2(2) (page 36) and 1.2.7(1) (page 31). The volume change for each element is then calculated, according to the equations of state, and the mass density is updated [6]. The volumetric information is needed for the calculation of stresses. With this information, the element stresses can be calculated [7] according to a relation between stresses and strains/strain rates, such as Eq. 1.2.8(1), page 31. The stresses are integrated over the elements, and the external loads are added to form the nodal forces F_n [8]. The nodal accelerations are then calculated [9] using

$$\ddot{D}_n = \frac{F_n}{m} + \frac{b}{\rho} \quad (3)$$

where b is the body force (Eq. 1.2.6(2), page 30), m is the nodal mass, and ρ is the mass density. The nodal velocities at $t_{n+\frac{1}{2}}$ are calculated [10] using Eq. (1) and the nodal displacements at t_{n+1} are calculated [11] using Eq. (2).

With explicit methods, a typical integration time step is about nanoseconds to microseconds; a typical simulation time is about 1 millisecond to 1 second, which will need many thousands or millions of cycles.

Explicit methods are useful for high-speed impact problems and highly nonlinear problems. For low-speed problems, where the durations are usually long, using explicit methods becomes impractical due to an enormous computing time, since it requires very small integration time steps. ↓



15.1.3 Solution Accuracy

[1] In **Transient Structural**, in which an implicit method is used, convergence criteria are used to control the solution accuracy, similar to the Newton-Raphson method described in 13.1.4, page 472. Equilibrium iterations imply that force balance must be satisfied. In **Explicit Dynamics**, since no equilibrium iterations are involved, solution accuracy is not controlled with convergence criteria. Instead, it uses the *principle of conservation of energy* to monitor the solution accuracy. It calculates overall energy at each cycle. If the energy error (to be defined) reaches a threshold, the solution is regarded as unstable and stops. The default threshold is 10% of a reference energy [2]. Energy statistics can be viewed by selecting **Energy Conservation** in **Solution Output** [3-4].

At any time, *Current Energy* of the system can be calculated, including its kinetic energy and strain energy. The *principle of work and energy*, a form of the principle of conservation of energy, states

$$(\text{Reference Energy}) + (\text{Work Done})_{\text{Reference} \rightarrow \text{Current}} = (\text{Current Energy}) \quad (1)$$

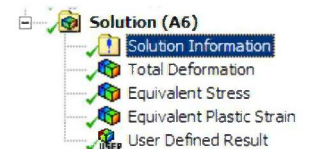
Where *Reference Energy* is the total energy of a reference time, default to the initial time. *Energy Error* is defined by

$$\text{Energy Error} = \frac{|(\text{Current Energy}) - (\text{Reference Energy}) - (\text{Work Done})_{\text{Reference} \rightarrow \text{Current}}|}{\max(|\text{Current Energy}|, |\text{Reference Energy}|, |\text{Kinetic Energy}|)} \quad (2) \quad \checkmark$$

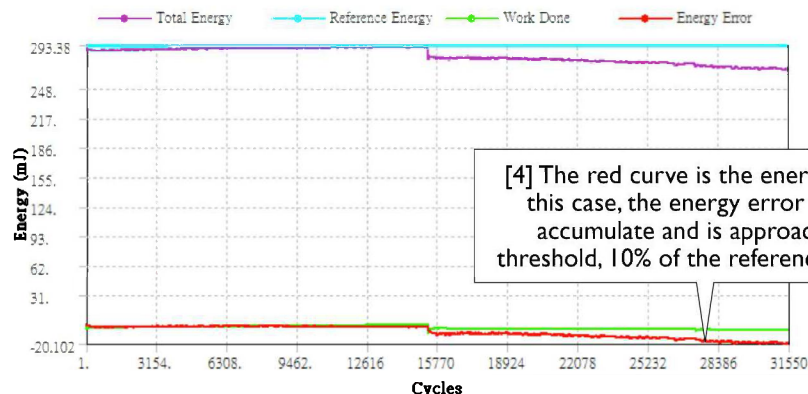
Details of "Analysis Settings"	
Analysis Settings Preference	
Type	Program Controlled
Step Controls	
Resume From Cycle	0
Maximum Number of Cycles	1e+07
End Time	5.e-004 s
Maximum Energy Error	0.1
Reference Energy Cycle	0
Initial Time Step	Program Controlled
Minimum Time Step	Program Controlled
Maximum Time Step	Program Controlled
Time Step Safety Factor	0.9
Characteristic Dimension	Diagonals
Automatic Mass Scaling	No

[2] If **Maximum Energy Error** reaches a threshold, the solution is regarded as unstable and stops. The default threshold is 0.1 (10% of a reference energy). ↓

[3] Energy statistics can be viewed by selecting **Energy Conservation** in **Solution Output**. ↓



Details of "Solution Information"	
Solution Information	
Solution Output	Energy Conservation
Update Interval	2.5 s
Display Points	All
Display Filter During Solve	Yes



15.1.4 Integration Time Steps

[1] With explicit methods, the integration time step needs to be small enough to ensure stability and accuracy of the solution. How small should the time step be? The German mathematicians, Courant, Friedrichs, and Lewy^[Refs 1, 2], suggested that, in a single time step Δt , a wave should not travel further than the smallest element size; i.e.,

$$\Delta t \leq \frac{h}{c} \quad (1)$$

where h is the smallest element size, c is the wave speed in the element. Eq. (1) is called the *CFL condition*. In **Explicit Dynamics**, a safety factor f is used to further ensure the solution stability [2-3]; i.e.,

$$\Delta t \leq f \frac{h}{c} \quad (2)$$

When generating meshes for **Explicit Dynamics**, you should make sure that a few very small elements do not control the time step, which is calculated according to the CFL condition. In general, a uniform mesh size is desirable in **Explicit Dynamics** simulations. ↘

Details of "Analysis Settings"	
Analysis Settings Preference	
Type	Program Controlled
Step Controls	
Resume From Cycle	0
Maximum Number of Cycles	1e+07
End Time	5.e-004 s
Maximum Energy Error	0.1
Reference Energy Cycle	0
Initial Time Step	Program Controlled
Minimum Time Step	Program Controlled
Maximum Time Step	Program Controlled
Time Step Safety Factor	0.9
Characteristic Dimension	Diagonals
Automatic Mass Scaling	No

[3] Workbench uses time step according to Eq. (2). #

[2] **Time Step Safety Factor** is used to further ensure the solution stability. It defaults to 0.9. ↑

15.1.5 Automatic Mass Scaling

[1] The wave speed in a material is $c = \sqrt{E/\rho}$, where E is the Young's modulus and ρ is the mass density of the material. Further, $\rho = m/V$, where m is the mass and V is the volume of an element. Substitution of these into Eq. 15.1.4(2) yields

$$\Delta t \leq fh \sqrt{\frac{m}{VE}} \quad (1)$$

The idea of mass scaling is to artificially increase the mass of small elements, so that the stability time step can be increased. Mass scaling is applied only to those elements which have a calculated stability time step less than a specified value, default to 1e-20 sec [2], which is to ensure that no mass scaling takes place. If a mesh contains very few small elements, this idea can be useful. Note that mass scaling changes the inertial properties of the model. Be careful to ensure that the model remains valid for the physical problem. →

Details of "Analysis Settings"	
Analysis Settings Preference	
Type	Custom
Step Controls	
Resume From Cycle	0
Maximum Number of Cycles	1e+07
End Time	5.e-004 s
Maximum Energy Error	0.1
Reference Energy Cycle	0
Initial Time Step	Program Controlled
Minimum Time Step	Program Controlled
Maximum Time Step	Program Controlled
Time Step Safety Factor	0.9
Characteristic Dimension	Diagonals
Automatic Mass Scaling	Yes
Minimum CFL Time Step	1.e-020 s
Maximum Element Scaling	100.
Maximum Part Scaling	5.e-002
Update Frequency	0

[2] **Automatic Mass Scaling** can be turned on. #

15.1.6 Static Damping

[1] **Explicit Dynamics** is designed for solving transient dynamic problems. To solve a static problem, we may perform a transient dynamic analysis and find the steady-state solution. **Static Damping** option [2] is to facilitate the finding of the steady-state solution. The idea is to introduce a damping force, to critically damp the lowest mode of oscillation.

The value of **Static Damping** for critical damping of the lowest mode of vibration is

$$\frac{2f\Delta t}{1 + 2\pi f\Delta t} \tag{1}$$

where f is the lowest frequency of the system.

Using the critical damping may minimize simulation run time. If **Static Damping** is larger than the critical damping, the solution would take unnecessarily longer time (than critical damping) to reach a steady state. On the other hand, if **Static Damping** is smaller than the critical damping, the solution would oscillate unnecessarily many times to reach a steady state.

For some highly-nonlinear static problems that fail with **Static Structural** analysis system, you may want to try this idea. →

Details of "Analysis Settings"	
⊞ Analysis Settings Preference	
⊞ Step Controls	
⊞ Solver Controls	
⊞ Euler Domain Controls	
⊞ Damping Controls	
Linear Artificial Viscosity	0.2
Quadratic Artificial Viscosity	1.
Linear Viscosity in Expansion	No
Hourglass Damping	AUTODYN Standard
Viscous Coefficient	0.1
Static Damping	0.

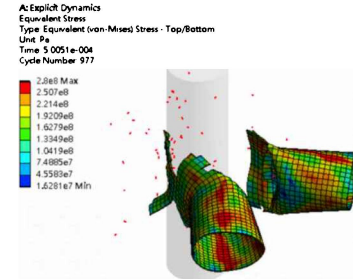
[2] **Static Damping** option can be used to solve a static problem. #

References

1. Wikipedia>Courant-Friedrichs-Lewy condition. (An English translation of the original paper can be downloaded from the webpage.)
2. Cook, R.D., Milkus, D. S., Plesha, M. E., and Witt, R. J., *Concepts and Applications of Finite Element Analysis, Fourth Edition*, John Wiley & Sons, Inc., 2002.

Section 15.2

High-Speed Impact

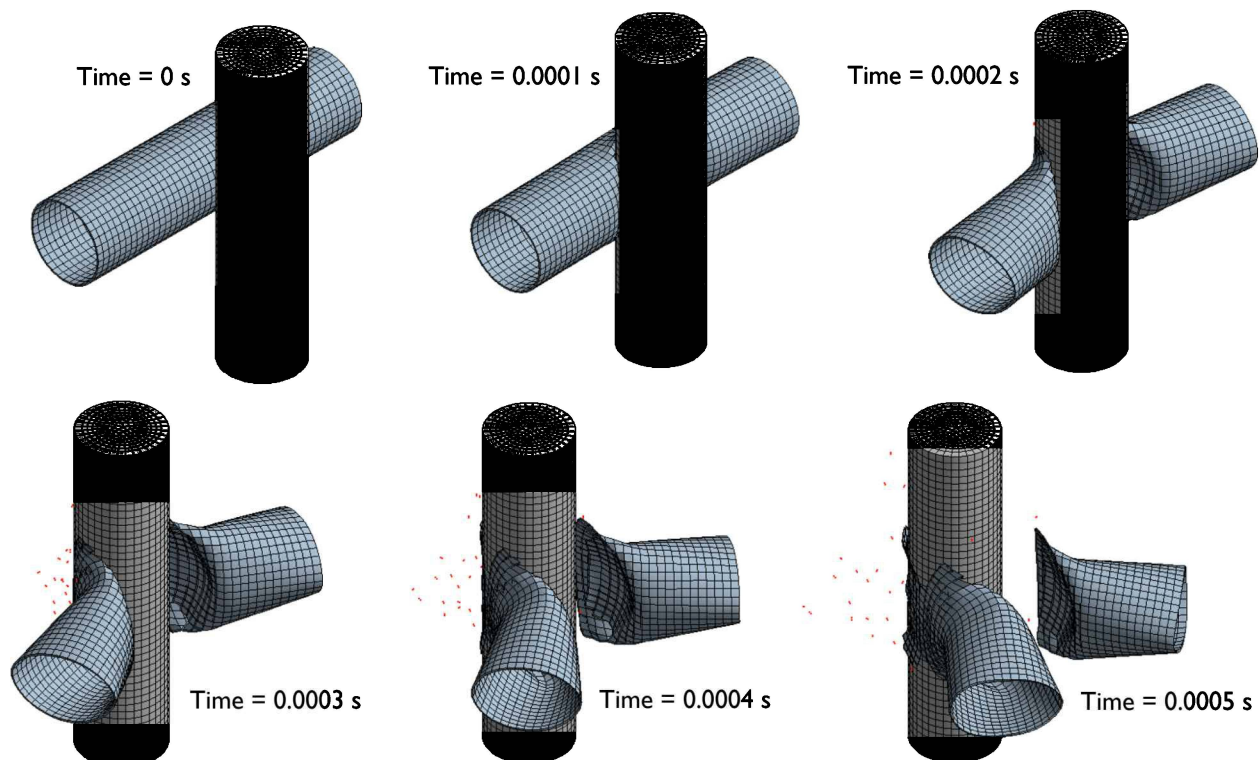


15.2.1 About the High-Speed Impact Simulation

[1] Imagine that, during an explosion, an aluminum pipe blasts away under the explosive pressure, hits a solid steel column, deforms, and is finally torn to fragments due to excessive strain (see snapshots below). In this section, we will simulate this scenario. We will use the default settings as much as possible to demonstrate that a complicated simulation like this can be done in **Explicit Dynamic** analysis system with just a few input data.

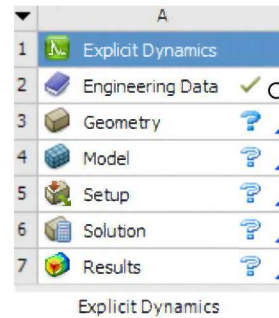
Both the aluminum pipe and the steel solid column have a diameter of 50 mm and a length of 200 mm. The steel column is modeled as a rigid body fixed in space. The aluminum pipe has a thickness of 1 mm and, right before hitting the pipe, has a speed of 300 m/s, about the speed of sound in the air. The aluminum is modeled as a bilinear isotropic plasticity material (Section 14.1) using the material parameters stored in **Engineering Data** with a modification that the tangent modulus is set to zero; i.e., the aluminum is modeled as a perfectly elastic-plastic material. It is assumed that the aluminum will be torn apart when the plastic strain is larger than 75%.

Millimeter will be used to create the geometry and the **SI** unit systems will be used in the simulation. #



15.2.2 Start Up

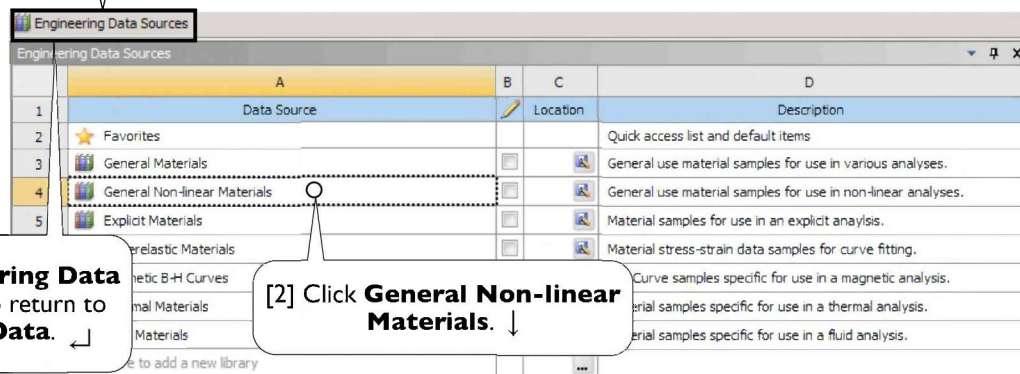
[1] Launch Workbench. Create an **Explicit Dynamics** analysis system by double-clicking it in **Toolbox**. Save the project as **Impact**. →



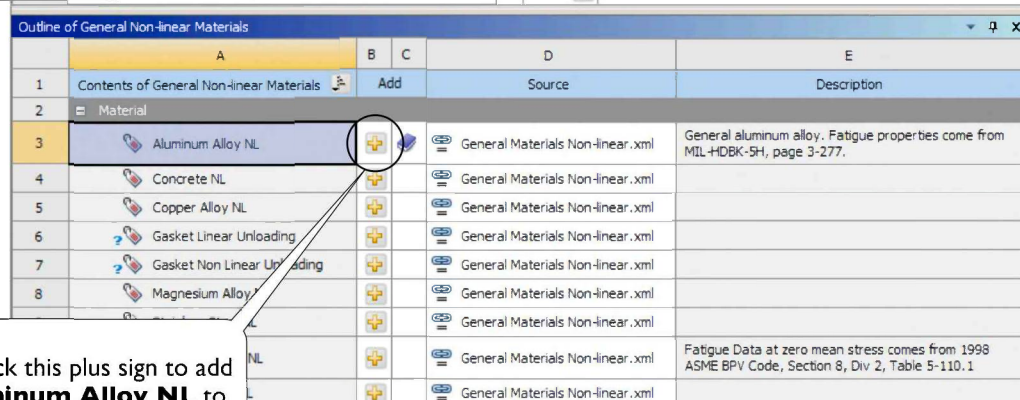
[2] Double-click **Engineering Data** to prepare material properties. #

15.2.3 Prepare Material Properties for Aluminum

[1] Click to switch to **Engineering Data Sources**. ↓



[4] Click **Engineering Data Sources** again to return to **Engineering Data**. ↩



[5, 7] Highlight **Aluminum Alloy NL**.

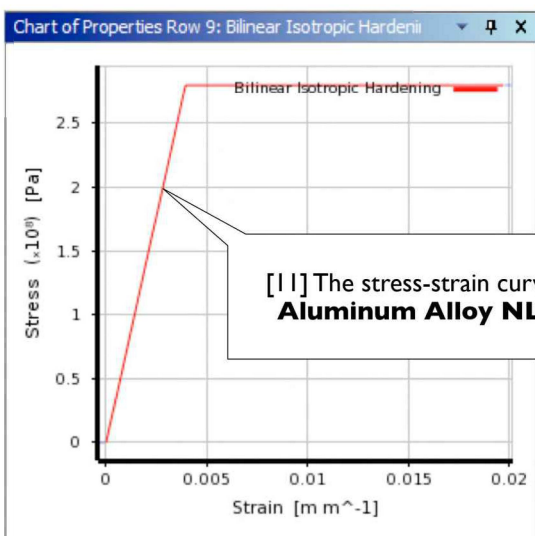
[8] Expand **Failure** and double-click **Plastic Strain Failure** to include this failure criterion. →

[10] Highlight **Bilinear Isotropic Hardening**. ↓

[9] Expand **Plastic Strain Failure** and type 0.75 for **Maximum Equivalent Plastic Strain EPS**. ←

[6] Expand **Bilinear Isotropic Hardening** and type 0 for **Tangent Modulus**. Note that the default unit system (**Metric**) is used here. ↑

Property	Value	Unit
Material Field Variables	Table	
Density	2770	kg m ⁻³
Isotropic Elasticity		
Derive from	Young's Modulus and Poisson's Ratio	
Young's Modulus	7.1E+10	Pa
Poisson's Ratio	0.33	
Bulk Modulus	6.9608E+10	Pa
Shear Modulus	2.6692E+10	Pa
Bilinear Isotropic Hardening		
Yield Strength	2.8E+08	Pa
Tangent Modulus	0	Pa
Specific Heat	875	J kg ⁻¹ C ⁻¹
Plastic Strain Failure		
Maximum Equivalent Plastic Strain EPS	0.75	



[12] Return to **Project Schematic**. #

15.2.4 Create Geometry

[1] Start up DesignModeler. ↓

[2] Select **Millimeter** as the length unit. In **ZXPlane**, draw a circle of diameter 50 mm. ↓

[3] Extrude the sketch 100 mm symmetrically both sides. ↗

[4] This is the steel column. ✓

[5] In **XYPlane**, draw a circle of diameter 50 mm like this. Remember to specify a horizontal distance of 50 mm. ↓

[6] Extrude the sketch to create a surface body. ↓

[7] This is the aluminum pipe. Close DesignModeler. #

Details View

Details of Extrude1

Extrude	Extrude1
Geometry	Sketch1
Operation	Add Material
Direction Vector	None (Normal)
Direction	Both - Symmetric
Extent Type	Fixed
FD1, Depth (>0)	100 mm
As Thin/Surface?	No
Merge Topology?	Yes

Geometry Selection: 1

Sketch	Sketch1
--------	---------

Generate

Details View

Details of Extrude2

Extrude	Extrude2
Geometry	Sketch2
Operation	Add Material
Direction Vector	None (Normal)
Direction	Both - Symmetric
Extent Type	Fixed
FD1, Depth (>0)	100 mm
As Thin/Surface?	Yes
FD2, Inward Thickness (>=0)	0 mm
FD3, Outward Thickness (>=0)	0 mm
Merge Topology?	Yes

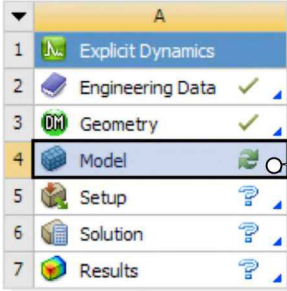
Geometry Selection: 1

Sketch	Sketch2
--------	---------

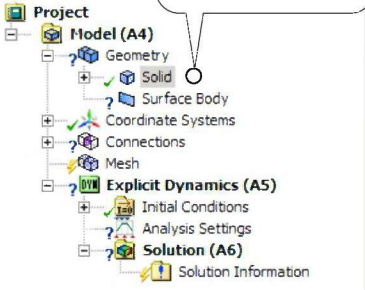
Generate

15.2.5 Set Up for Simulation

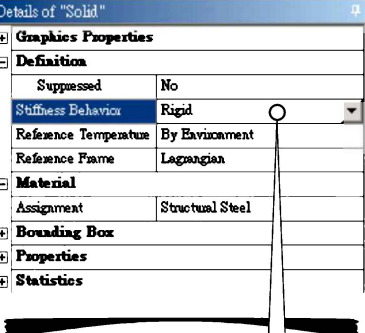
[1] Start up **Mechanical**. Select the **m-kg-N-s** unit system. →



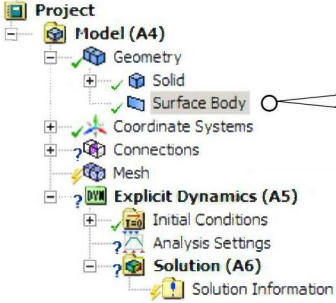
[2] Highlight **Solid**. ↓



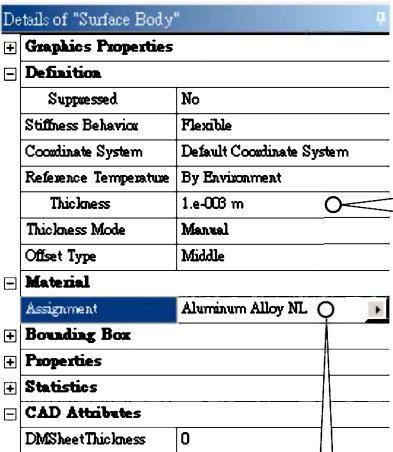
[3] Select **Rigid** for **Stiffness Behavior**. ←



[4] Highlight **Surface Body**. ↓

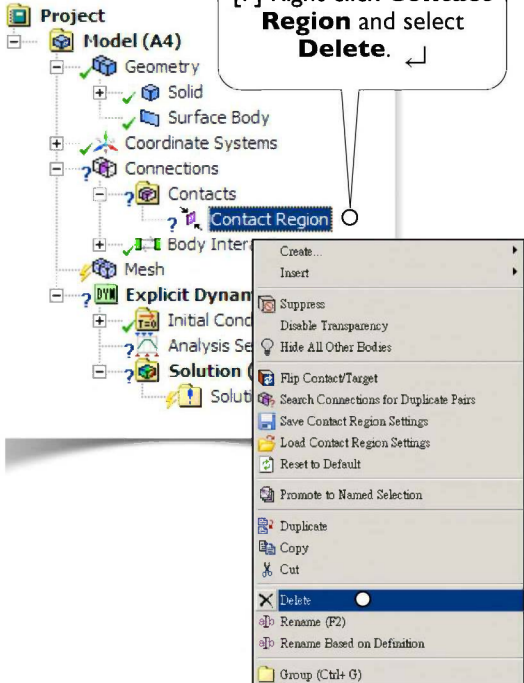


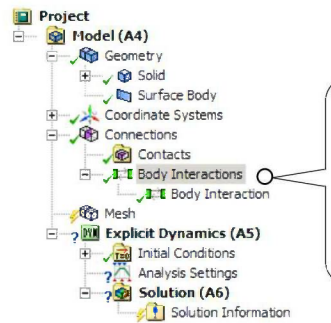
[5] Type 0.001 (m) for **Thickness**. ↓



[6] Select **Aluminum Alloy NL**. ↗

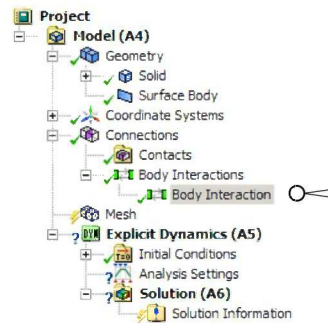
[7] Right-click **Contact Region** and select **Delete**. ↶





[8] Highlight **Body Interactions** (the one with an "s") to view the details and leave all settings their defaults. →

Details of "Body Interactions"	
Advanced	
Contact Detection	Trajectory
Formulation	Penalty
Sliding Contact	Discrete Surface
Shell Thickness Factor	0.
Node Shell Thickness	
Body Self Contact	Program Controlled
Element Self Contact	Program Controlled
Tolerance	0.2



[9] Highlight **Body Interaction** (the one without an "s") to view the details and leave all settings their default. ↓

Details of "Body Interaction"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Frictionless
Suppressed	No

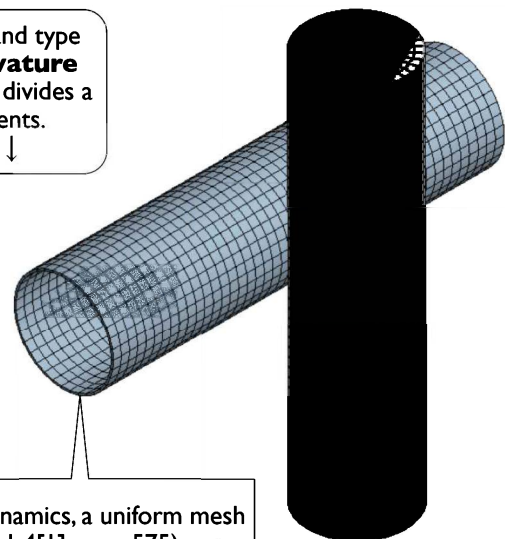
Contacts vs. Body Interactions?

[10] **Body Interactions** is to specify contacts between bodies while **Contacts** is to specify contacts between surfaces. You can choose either way to specify the contact relations. **Body Interactions** is simpler, but **Contacts** may be more computationally efficient. By default, **Frictionless** body interactions are established among all bodies.

A feature of **Body Interactions** is that two bodies can be specified as both **Bonded** and **Frictionless** (or **Frictional**). In that case, two bodies are bonded initially. After the bond breaks during the simulation, the frictionless (or frictional) contact will take place. ✓

Details of "Mesh"	
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Explicit
Element Order	Linear
<input type="checkbox"/> Element Size	Default (1.6148e-002 m)
Sizing	
Use Adaptive Sizing	No
Use Uniform Size Function For Sheets	No
<input type="checkbox"/> Growth Rate	Default (1.2)
<input type="checkbox"/> Max Size	Default (1.6148e-002 m)
Mesh Defeatureing	Yes
<input type="checkbox"/> Defeature Size	Default (8.0742e-005 m)
Capture Curvature	Yes
<input type="checkbox"/> Curvature Min Size	Default (1.6148e-004 m)
<input checked="" type="checkbox"/> Curvature Normal Angle	10.0°
Capture Proximity	No
Bounding Box Diagonal	0.3 m
Average Surface Area	1.6689e-002 m²
Minimum Edge Length	0.15708 m
Quality	
Check Mesh Quality	Yes, Errors
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	High
Mesh Metric	Skewness
<input type="checkbox"/> Min	4.5121e-006
<input type="checkbox"/> Max	0.43419
<input type="checkbox"/> Average	0.11476
<input type="checkbox"/> Standard Deviation	9.1233e-002
Inflation	
Advanced	
Statistics	
<input type="checkbox"/> Nodes	13226
<input type="checkbox"/> Elements	12310

[11] Highlight **Mesh** and type 10 (degrees) for **Curvature Normal Angle**. This divides a circle into 36 elements. Generate mesh. ↓

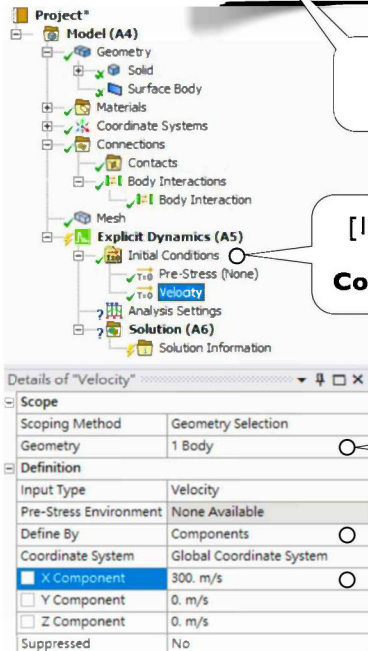


[12] In explicit dynamics, a uniform mesh is desirable (15.1.4[1], page 575). ↩



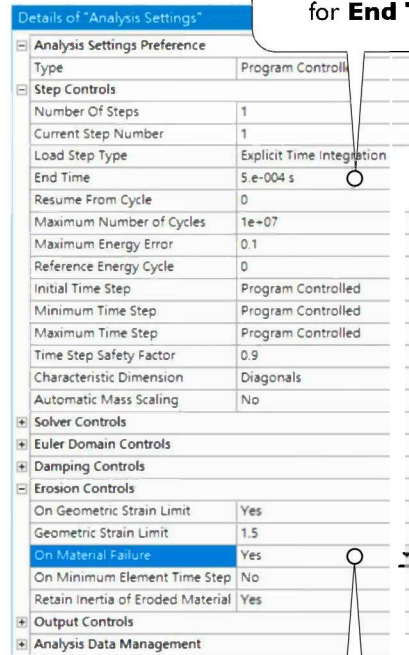
[14] Click **Velocity**. ↓

[13] Highlight **Initial Conditions**. ↑



[15] Select the surface body (aluminum pipe), using body filter. ↗

[16] Highlight **Analysis Settings**. Type 0.0005 (s) for **End Time**. ↓

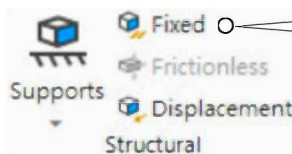


[17] Turn on **On Material Failure** (see [18]). ↓

Erosion Controls^[Ref 1]

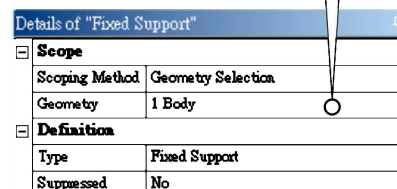
[18] **Erosion Controls** in **Analysis Settings** determines the conditions under which an element will be removed. The default condition is that an element is removed when its *geometric strain*, or *effective strain*^[Ref 1], exceeds a limit of 150%. This value is large enough to assure that no elements are removed by default.

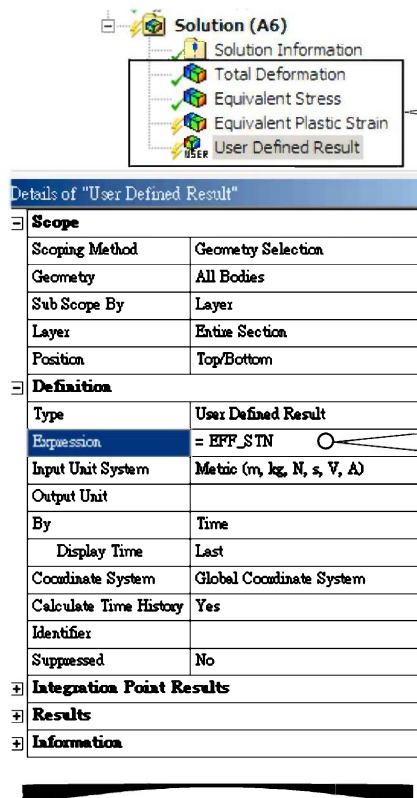
In this case, we add another failure condition: an element is removed when its plastic strain exceeds 75% (see [17] and 15.2.3[9], page 579). ↓



[19] Insert a **Fixed Support**. →

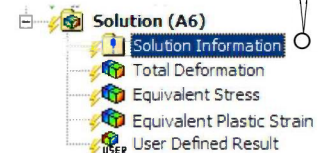
[20] Select the solid body (the steel column). ↶





[21] Insert result objects like this. Note that the last object is a **User Defined Result** ([22]; also see 3.3.10[1-2], page 148). ↓

[23] Highlight **Solution Information**. ↓



Details of "Solution Information"

Solution Information	
Solution Output	Solver Output
Update Interval	2.5 s
Display Points	All
Display Filter During Solve	Yes



[24] Solve. ✓

```

Cycle: 963, Time: 4.933E-04s, Time Inc.: 5.135E-07s, Progress: 98.66%, Est. Clock Time Remaining: 0s
Cycle: 964, Time: 4.938E-04s, Time Inc.: 5.135E-07s, Progress: 98.77%, Est. Clock Time Remaining: 0s
Cycle: 965, Time: 4.943E-04s, Time Inc.: 5.135E-07s, Progress: 98.87%, Est. Clock Time Remaining: 0s
Cycle: 966, Time: 4.949E-04s, Time Inc.: 5.135E-07s, Progress: 98.97%, Est. Clock Time Remaining: 0s
Cycle: 967, Time: 4.954E-04s, Time Inc.: 5.135E-07s, Progress: 99.07%, Est. Clock Time Remaining: 0s
Cycle: 968, Time: 4.959E-04s, Time Inc.: 5.135E-07s, Progress: 99.18%, Est. Clock Time Remaining: 0s
Cycle: 969, Time: 4.964E-04s, Time Inc.: 5.134E-07s, Progress: 99.28%, Est. Clock Time Remaining: 0s
Cycle: 970, Time: 4.969E-04s, Time Inc.: 5.134E-07s, Progress: 99.38%, Est. Clock Time Remaining: 0s
Cycle: 971, Time: 4.974E-04s, Time Inc.: 5.134E-07s, Progress: 99.49%, Est. Clock Time Remaining: 0s
Cycle: 972, Time: 4.979E-04s, Time Inc.: 5.134E-07s, Progress: 99.59%, Est. Clock Time Remaining: 0s
Cycle: 973, Time: 4.985E-04s, Time Inc.: 5.134E-07s, Progress: 99.69%, Est. Clock Time Remaining: 0s
Cycle: 974, Time: 4.990E-04s, Time Inc.: 5.134E-07s, Progress: 99.79%, Est. Clock Time Remaining: 0s
Cycle: 975, Time: 4.995E-04s, Time Inc.: 5.134E-07s, Progress: 99.90%, Est. Clock Time Remaining: 0s
Cycle: 976, Time: 5.000E-04s, Time Inc.: 5.134E-07s, Progress: 100.00%, Est. Clock Time Remaining: 0s
Cycle: 977, Time: 5.005E-04s, Time Inc.: 5.134E-07s, Progress: 100.00%, Est. Clock Time Remaining: -
  
```

[25] It takes about 1000 cycles to complete the simulation. #

15.2.6 Animate the Deformation

[1] Highlight **Total Deformation**. →

[2] Select **Contours/Solid Fill**. ↓

[3] Select **10 Sec.** ←

[4] Select **Result Sets**. ←

[5] Click **Play**. #

Details of "Total Deformation"

Scoping Method	Geometry Selection
Geometry	All Bodies

Definition

Type	Total Deformation
By	Time
Display Time	Last
Calculate Time History	Yes
Identifier	
Suppressed	No

Results

Minimum	0. m
Maximum	0.16256 m

Graph

Animation | Frames | 10 Sec

0.16256
0.1
0.05
0
-0.05
-0.1

0 1e-4 2e-4 3e-4 4e-4 5.0051e-4

[s]

1

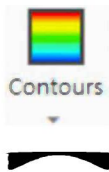
Graphics Annotations | Graph

Tabular Data

Time [s]	Minimum [m]
1	1.1755e-038 0.
2	2.5511e-005 0.
3	5.0086e-005 0.
4	7.5152e-005 0.
5	1.0003e-004 0.
6	1.253e-004 0.
7	1.5002e-004 0.
8	1.7518e-004 0.
9	2.0034e-004 0.
10	2.2505e-004 0.
11	2.5025e-004 0.

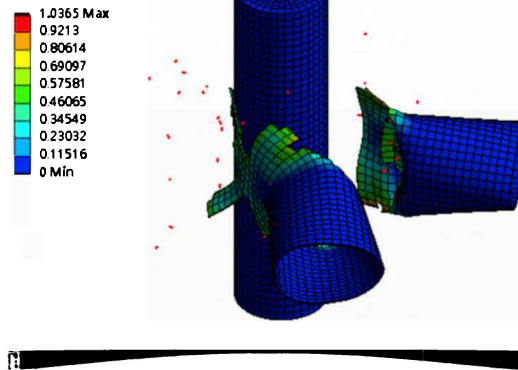
Ready | Messages pane | No Selection | Metric (m, kg, N, s, V, A) | Degrees | RPM | Celsius

15.2.7 View Numerical Results

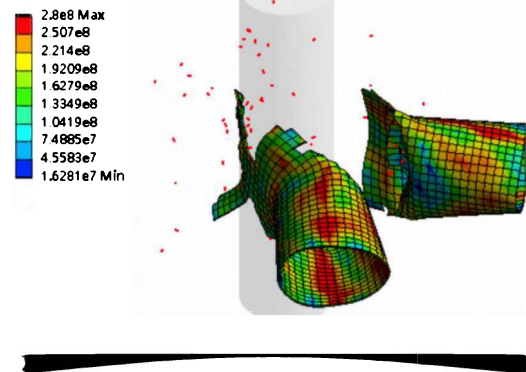


[1] Select **Contours/Smooth Contours** and examine the results.

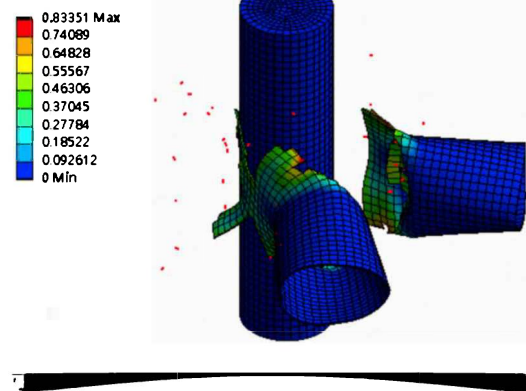
A: Explicit Dynamics
Equivalent Plastic Strain
Type: Equivalent Plastic Strain - Top/Bottom
Unit: m/m
Time: 5.0051e-004
Cycle Number: 977



A: Explicit Dynamics
Equivalent Stress
Type: Equivalent (von-Mises) Stress - Top/Bottom
Unit: Pa
Time: 5.0051e-004
Cycle Number: 977



A: Explicit Dynamics
User Defined Result
Expression: EFF-STN
Time: 5.0051e-004
Cycle Number: 977



Wrap Up

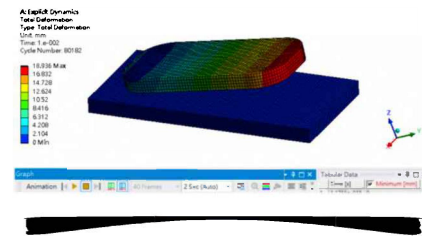
[2] Save the project and exit Workbench. #

Reference

1. All Help>Mechanical Application>Explicit Dynamics Analysis Guide>Explicit Dynamics Theory Guide>Analysis Settings>Erosion Controls

Section 15.3

Drop Test

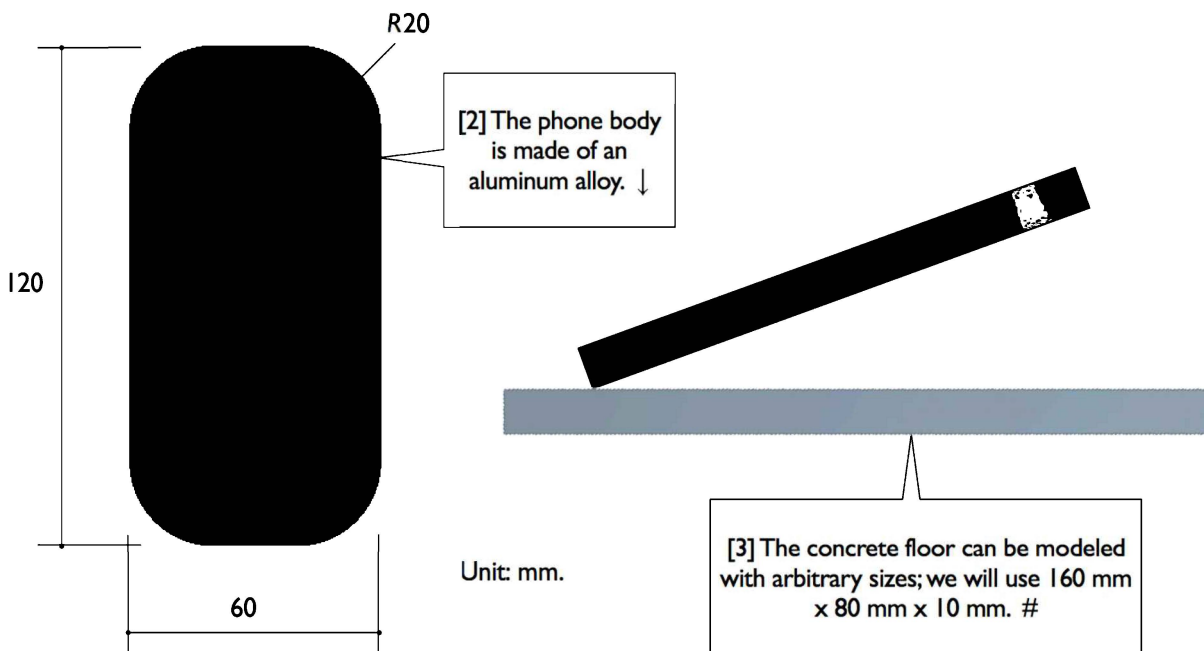


15.3.1 About the Drop Test Simulation

[1] Drop test simulation is a special case of impact simulation, in which one of the impacting objects is a stationary floor, typically made of concrete, steel, or stone. In this section, we consider a scenario that a mobile phone falls out of your pocket and drops on a concrete floor. This kind of simulation typically takes hours of computing time. We learned from Section 15.2 that a typical integration time step in **Explicit Dynamics** is 10^{-7} to 10^{-8} seconds. It would take about 100,000 to 1,000,000 cycles to complete a 0.01 seconds drop test. In this section, we will simplify the model to minimize the run time. A more realistic model will be suggested and left as an exercise (15.4.2, page 600).

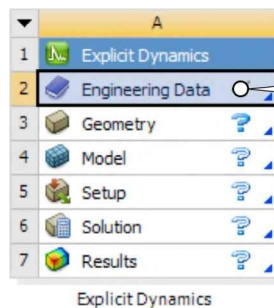
The phone body is a shell of thickness 0.5 mm and made of an aluminum alloy [2]. The concrete floor is modeled as an 160 mm x 80 mm x 10 mm block [3]. When the phone hits the floor, its velocity is 5 m/s, which is equivalent to a free fall from a height of 1.25 m. We will assume that the phone body forms an angle of 20° with the horizon when it hits the floor.

We will use **mm-kg-N-s** unit system in the simulation. ↓



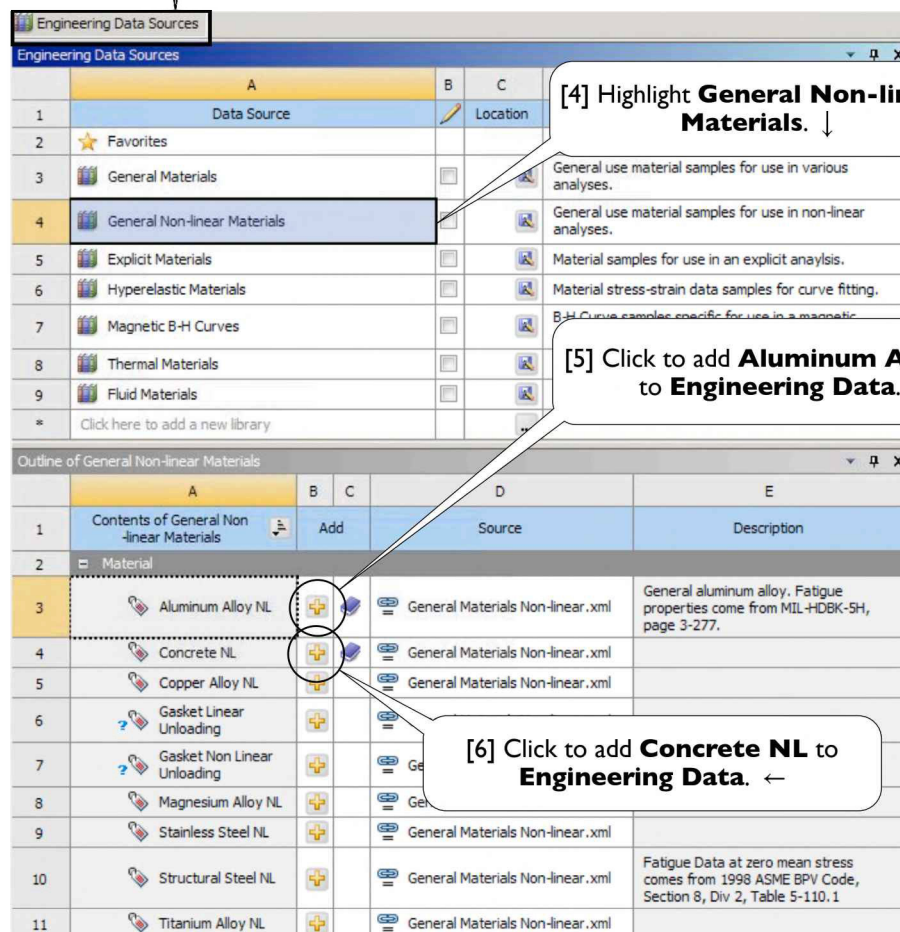
15.3.2 Start Up

[1] Launch Workbench. Create an **Explicit Dynamics** analysis system by double-clicking it in **Toolbox**. Save the project as **Drop**. ↓



[2] Double-click **Engineering Data** to prepare material properties. ↓

[3] Click to switch to **Engineering Data Sources**. ↓

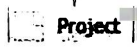


[4] Highlight **General Non-linear Materials**. ↓

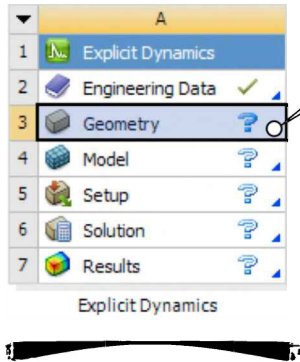
[5] Click to add **Aluminum Alloy NL** to **Engineering Data**. ↓

[6] Click to add **Concrete NL** to **Engineering Data**. ←

[7] Return to **Project Schematic**. #

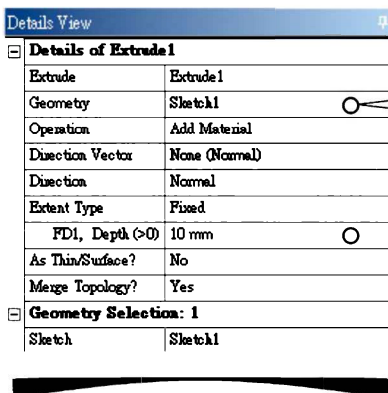
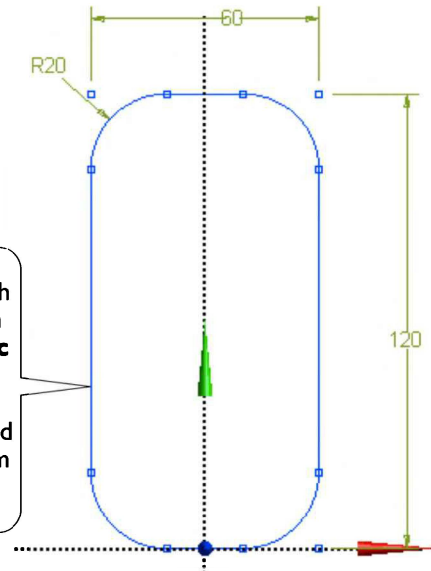


15.3.3 Create Geometry

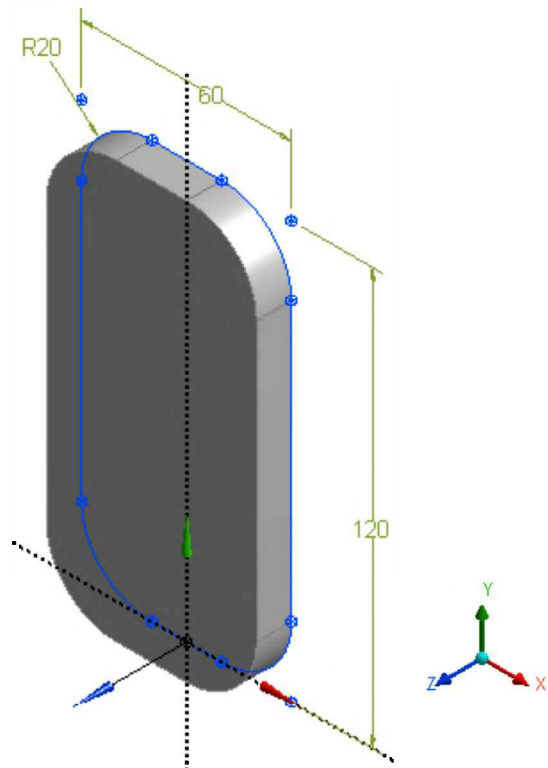


[1] Start up DesignModeler. ↓

[2] Select **Millimeter** as the length unit. On **XYPlane**, draw a sketch like this. The sketch is **Symmetric** about the Y-axis and the bottom edge is **Coincident** with the X-axis. (To make all the edges blue, add a **Coincident** between the bottom edge and the X-axis.) ↓



[3] Extrude the sketch to create the phone body. ↵

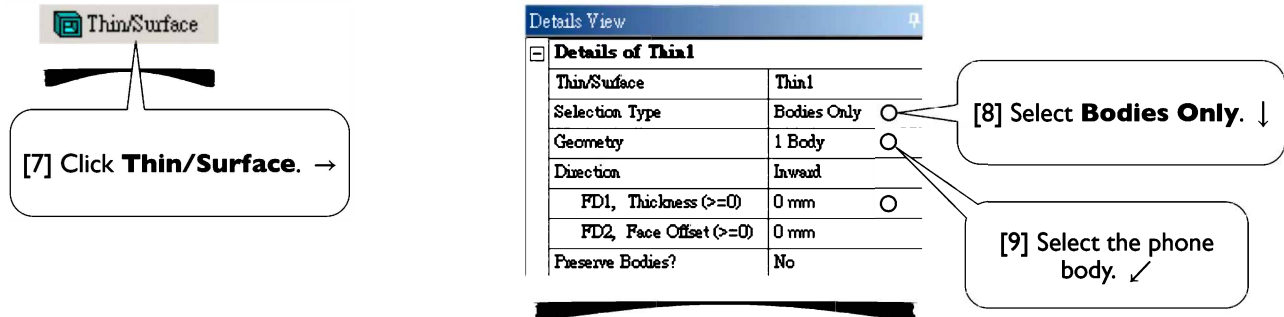


[4] In **YZPlane**, draw a rectangle using **Rectangle by 3 Points** tool. Specify dimensions, including the angle. Note that the rectangle is not blue-colored yet. ↓

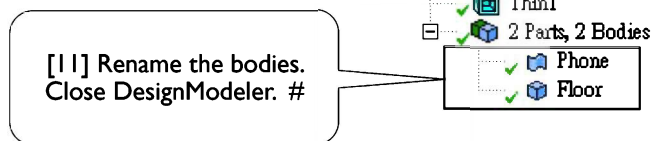
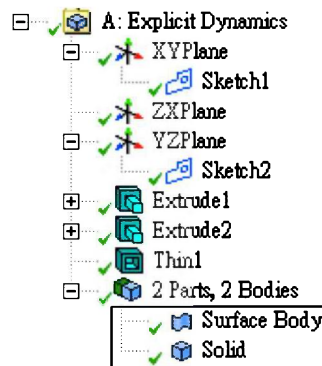
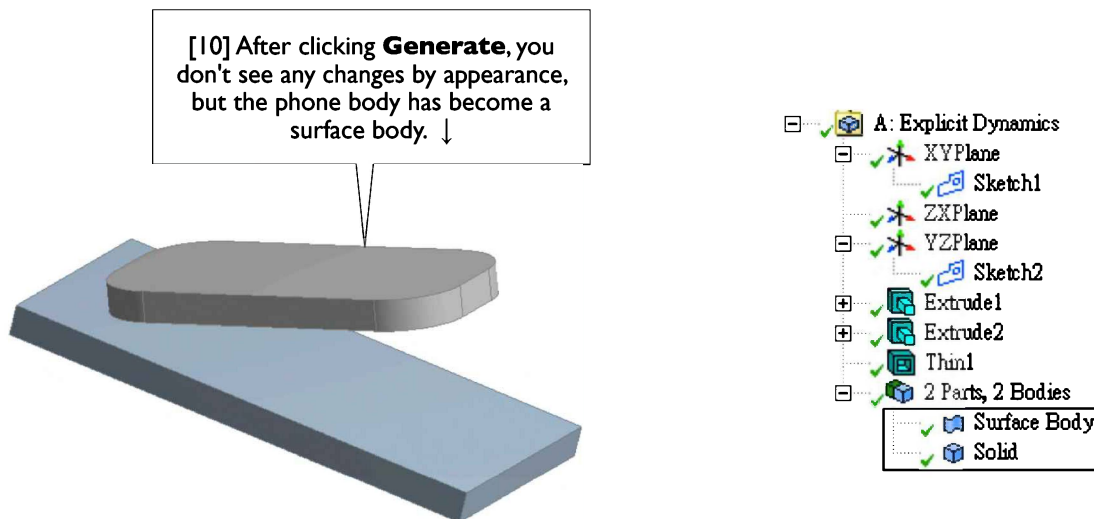
[5] Impose a constraint so that the origin is **Coincident** with an edge of the rectangle as shown. When you select the origin, you may need to turn off the edge filter and leave the point filter on. Also specify the dimension (20 mm) using the **Length/Distance** tool. ↓

[6] Extrude the sketch 40 mm both sides to create a concrete block. ↵

Details View	
Details of Extrude2	
Extrude	Extrude2
Geometry	Sketch2
Operation	Add Material
Direction Vector	None (Normal)
Direction	Both - Symmetric
Extent Type	Fixed
FD1, Depth (>0)	40 mm
As Thin/Surface?	No
Merge Topology?	Yes
Geometry Selection: 1	
Sketch	Sketch2



Generate



15.3.4 Set Up for Simulation

[1] Start up **Mechanical**. Select the **mm-kg-N-s** unit system. ←

[2] Highlight **Phone**. ↓

[3] Type 0.5 (mm) for **Thickness** and assign the material. ↗

[4] Highlight **Floor**. ↓

[5] Select **Rigid** and assign the material. ✓

[6] With **Coordinate Systems** in the project tree highlighted, click **Coordinate System**. →

[7] A **Coordinate System** is created. This coordinate system will be used to specify the earth gravity and the initial velocity. ↶

Details of "Phone"

Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Thickness	0.5 mm
Thickness Mode	Manual
Offset Type	Middle
Material	
Assignment	Aluminum Alloy NL
Bounding Box	
Properties	
Statistics	
CAD Attributes	
DMSheetThickness	0

Details of "Floor"

Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Rigid
Reference Temperature	By Environment
Reference Frame	Lagrangian
Material	
Assignment	Concrete NL
Bounding Box	
Properties	
Statistics	

Details of "Coordinate System"	
Definition	
Type	Cartesian
Suppressed	No
Origin	
Define By	Global Coordinates <input type="radio"/>
Origin X	0. mm
Origin Y	0. mm
Origin Z	0. mm
Location	Click to Change
Principal Axis	
Axis	Z <input type="radio"/>
Define By	Geometry Selection <input type="radio"/>
Geometry	Click to Change <input type="radio"/>
Orientation About Principal Axis	
Axis	Y <input type="radio"/>
Define By	Geometry Selection <input type="radio"/>
Geometry	Click to Change <input type="radio"/>
Directional Vectors	
Transformations	
Base Configuration	Absolute
Transformed Configuration	[0. 0. 0.]

[8] Choose any edge parallel to the thickness direction of the concrete block (as shown in [9-10]) to define the Z-axis. ↓

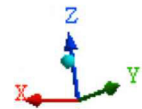
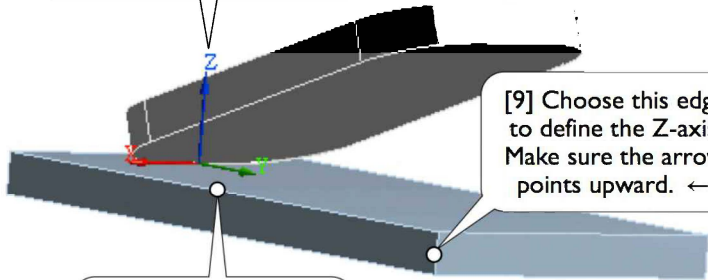
[11] Choose any edge parallel to the longest direction of the concrete block (as shown in [12-13]) to define the Y-axis. ↓

[14] Make sure the newly created coordinate system is like this. ↘

[9] Choose this edge to define the Z-axis. Make sure the arrow points upward. ←

[12] Choose this edge to define the Y-axis. Make sure the arrow points rightward. ←

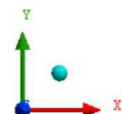
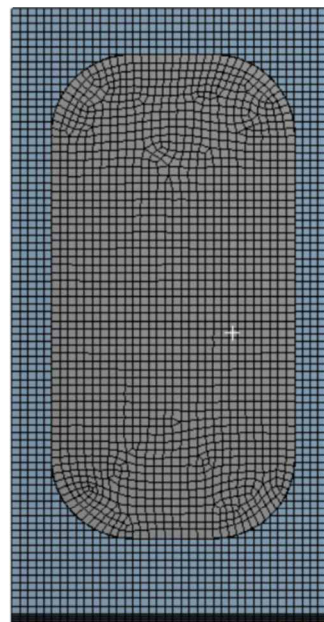
[10, 13] You may need to click this button so that the arrow points to the correct direction. ↗ ↗

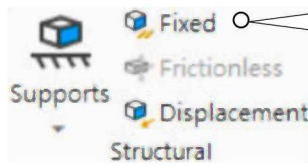


[15] This is the global coordinate system. ←

Details of "Mesh"	
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Explicit
Element Order	Linear
<input type="checkbox"/> Element Size	2.0 mm
Sizing	
Quality	
Inflation	
Advanced	
Statistics	
<input type="checkbox"/> Nodes	24292
<input type="checkbox"/> Elements	20372

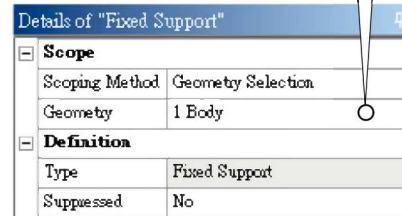
[16] Set the element size to 2 mm (the default is 6.8 mm) and generate the mesh. ↵





[17] With **Explicit Dynamics** highlighted, insert a **Fixed Support**. →

[18] Select the concrete body (using the body filter). ✓



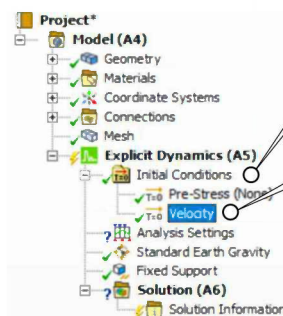
[19] Insert an **Inertial/Standard Earth Gravity**. ↓



[23] Click **Velocity**. ↘

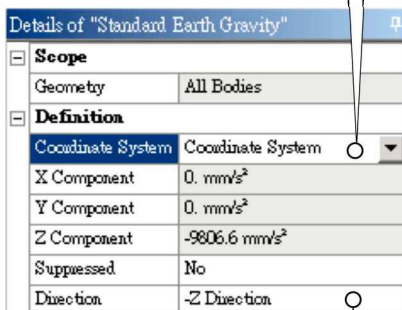
[22] Highlight **Initial Conditions**. ←

[24] A **Velocity** is inserted. ↓

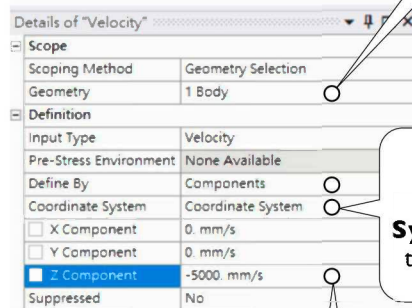


[25] Select the phone body. ↓

[20] Select **Coordinate System**, which was created in steps [6-14] (pages 592-593). ↓

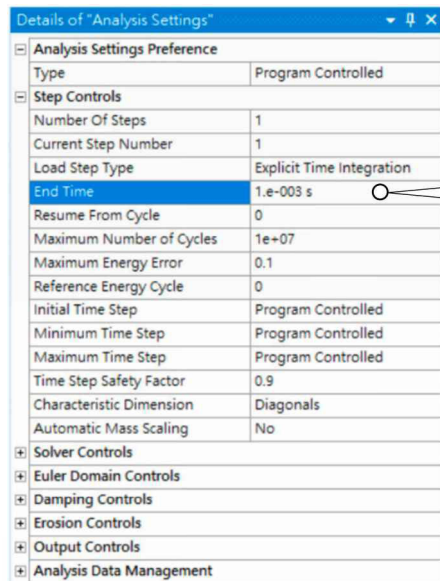


[21] **-Z Direction** is the default direction. ↗



[26] Select **Coordinate System**, the same as that used in [20]. ↓

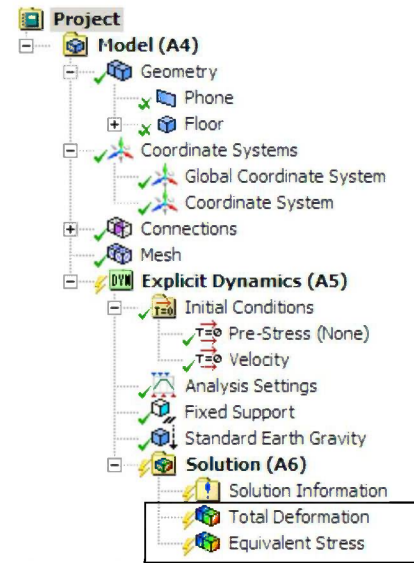
[27] Type -5000 (mm/s) for **Z Component**. ↙



[28] With **Analysis Settings** highlighted, type 0.001 (s) for **End Time**. ↘



[30] Highlight **Solution Information** and solve the model. ↓



[29] Insert a **Total Deformation** and an **Equivalent Stress**. ←

```

Cycle:      8009, Time:    9.984E-04s, Time Inc.: 1.247E-07s, Progress: 99.84%, Est. Clock Time Remaining: 0s
Cycle:      8010, Time:    9.985E-04s, Time Inc.: 1.247E-07s, Progress: 99.85%, Est. Clock Time Remaining: 0s
Cycle:      8011, Time:    9.986E-04s, Time Inc.: 1.247E-07s, Progress: 99.86%, Est. Clock Time Remaining: 0s
Cycle:      8012, Time:    9.987E-04s, Time Inc.: 1.247E-07s, Progress: 99.87%, Est. Clock Time Remaining: 0s
Cycle:      8013, Time:    9.989E-04s, Time Inc.: 1.247E-07s, Progress: 99.89%, Est. Clock Time Remaining: 0s
Cycle:      8014, Time:    9.990E-04s, Time Inc.: 1.247E-07s, Progress: 99.90%, Est. Clock Time Remaining: 0s
Cycle:      8015, Time:    9.991E-04s, Time Inc.: 1.247E-07s, Progress: 99.91%, Est. Clock Time Remaining: 0s
Cycle:      8016, Time:    9.992E-04s, Time Inc.: 1.247E-07s, Progress: 99.92%, Est. Clock Time Remaining: 0s
Cycle:      8017, Time:    9.994E-04s, Time Inc.: 1.247E-07s, Progress: 99.94%, Est. Clock Time Remaining: 0s
Cycle:      8018, Time:    9.995E-04s, Time Inc.: 1.247E-07s, Progress: 99.95%, Est. Clock Time Remaining: 0s
Cycle:      8019, Time:    9.996E-04s, Time Inc.: 1.247E-07s, Progress: 99.96%, Est. Clock Time Remaining: 0s
Cycle:      8020, Time:    9.997E-04s, Time Inc.: 1.247E-07s, Progress: 99.97%, Est. Clock Time Remaining: 0s
Cycle:      8021, Time:    9.999E-04s, Time Inc.: 1.247E-07s, Progress: 99.99%, Est. Clock Time Remaining: 0s
Cycle:      8022, Time:    1.000E-03s, Time Inc.: 1.247E-07s, Progress: 100.00%, Est. Clock Time Remaining: 0s
Cycle:      8023, Time:    1.000E-03s, Time Inc.: 1.247E-07s, Progress: 100.00%, Est. Clock Time Remaining: -
  
```

[31] Each cycle increases about 0.125 microseconds. ↓

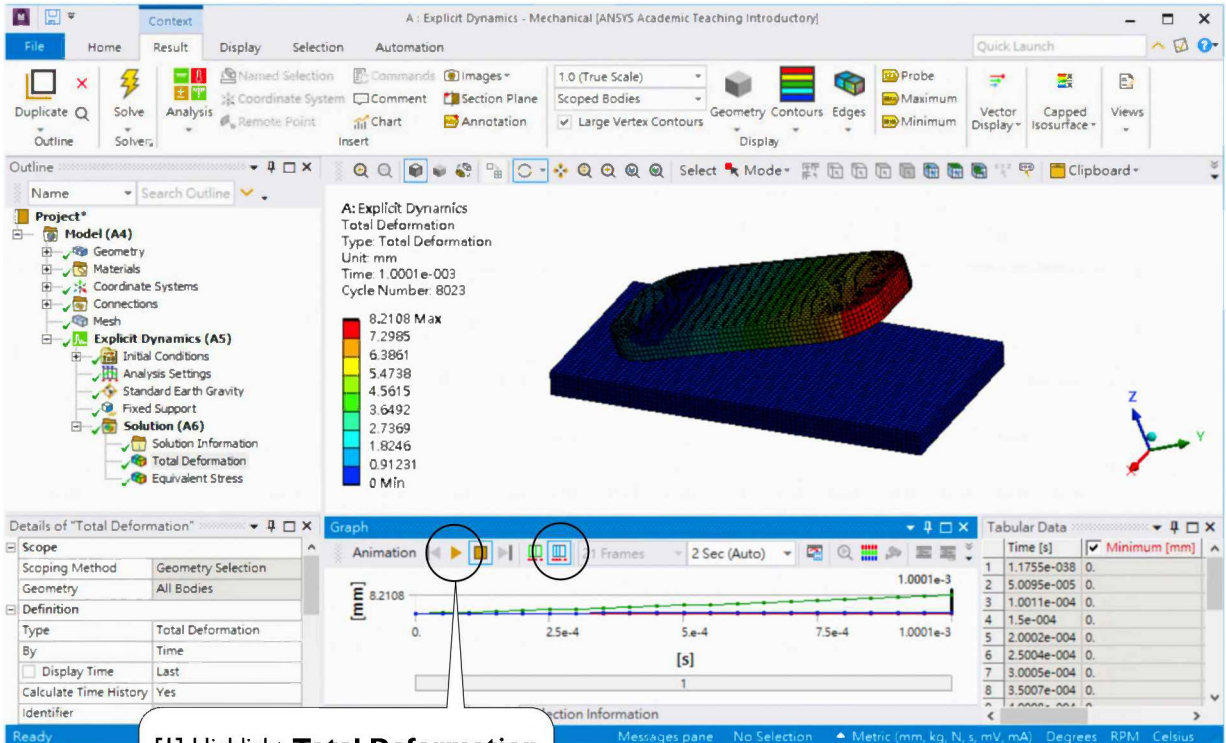
Integration Time Steps

[32] In this case, the integration time step is about 0.125 microseconds. This number, proportional to the size of the smallest element, controls the overall run time (15.1.4, page 575). In **Explicit Dynamics**, a mesh of uniform element size is the most efficient mesh.

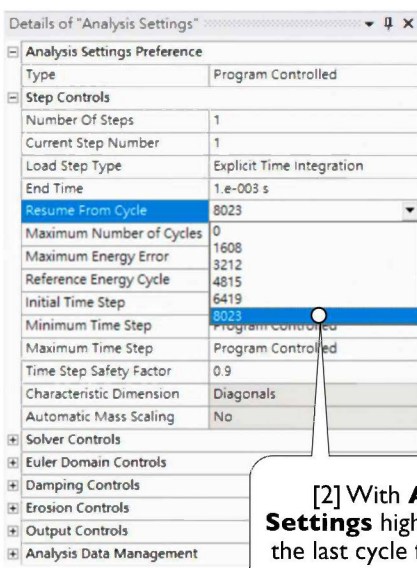
Perform Simulation Incrementally

A drop test simulation typically takes hours of computing time. If you make a mistake, it will waste a significant amount of time. As a good practice, always perform the simulation incrementally. In this case, we perform a 0.001 seconds simulation first to see if anything goes wrong. If everything is okay, then we can continue the simulation starting from the last cycle. This feature is called the **Restart** of the simulation. #

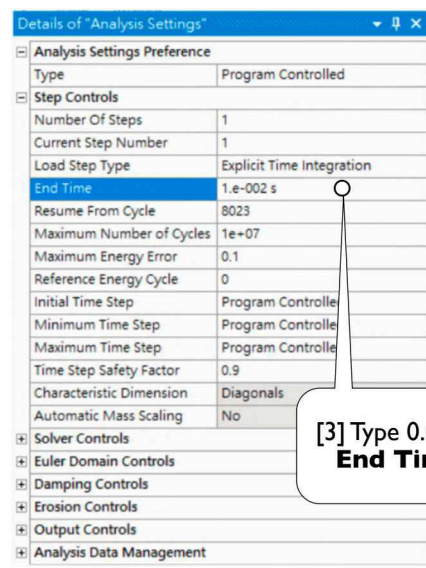
15.3.5 View the Results



[1] Highlight **Total Deformation** and animate the results. The results look reasonable. Let's extend the simulation time. ↓



[2] With **Analysis Settings** highlighted, select the last cycle for **Resume From Cycle**. →



[3] Type 0.01 (s) for **End Time**. ↵



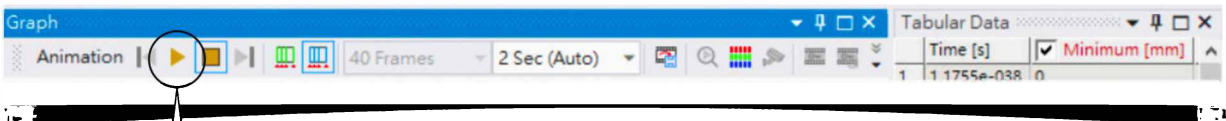
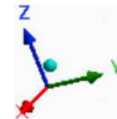
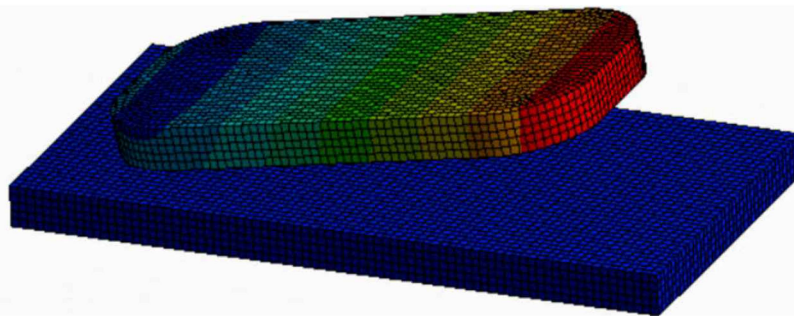
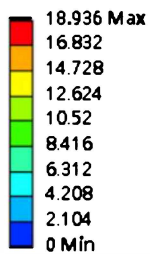
[4] Highlight **Solution Information** and solve again. It takes a while. ↓

```

Cycle: 80168, Time: 9.998E-03s, Time Inc.: 1.247E-07s, Progress: 99.98%, Est. Clock Time Remaining: 0s
Cycle: 80169, Time: 9.998E-03s, Time Inc.: 1.247E-07s, Progress: 99.98%, Est. Clock Time Remaining: 0s
Cycle: 80170, Time: 9.999E-03s, Time Inc.: 1.247E-07s, Progress: 99.98%, Est. Clock Time Remaining: 0s
Cycle: 80171, Time: 9.999E-03s, Time Inc.: 1.247E-07s, Progress: 99.99%, Est. Clock Time Remaining: 0s
Cycle: 80172, Time: 9.999E-03s, Time Inc.: 1.247E-07s, Progress: 99.99%, Est. Clock Time Remaining: 0s
Cycle: 80173, Time: 9.999E-03s, Time Inc.: 1.247E-07s, Progress: 99.99%, Est. Clock Time Remaining: 0s
Cycle: 80174, Time: 9.999E-03s, Time Inc.: 1.247E-07s, Progress: 99.99%, Est. Clock Time Remaining: 0s
Cycle: 80175, Time: 9.999E-03s, Time Inc.: 1.247E-07s, Progress: 99.99%, Est. Clock Time Remaining: 0s
Cycle: 80176, Time: 9.999E-03s, Time Inc.: 1.247E-07s, Progress: 99.99%, Est. Clock Time Remaining: 0s
Cycle: 80177, Time: 9.999E-03s, Time Inc.: 1.247E-07s, Progress: 99.99%, Est. Clock Time Remaining: 0s
Cycle: 80178, Time: 1.000E-02s, Time Inc.: 1.247E-07s, Progress: 100.00%, Est. Clock Time Remaining: 0s
Cycle: 80179, Time: 1.000E-02s, Time Inc.: 1.247E-07s, Progress: 100.00%, Est. Clock Time Remaining: 0s
Cycle: 80180, Time: 1.000E-02s, Time Inc.: 1.247E-07s, Progress: 100.00%, Est. Clock Time Remaining: 0s
Cycle: 80181, Time: 1.000E-02s, Time Inc.: 1.247E-07s, Progress: 100.00%, Est. Clock Time Remaining: 0s
Cycle: 80182, Time: 1.000E-02s, Time Inc.: 1.247E-07s, Progress: 100.00%, Est. Clock Time Remaining: -

```

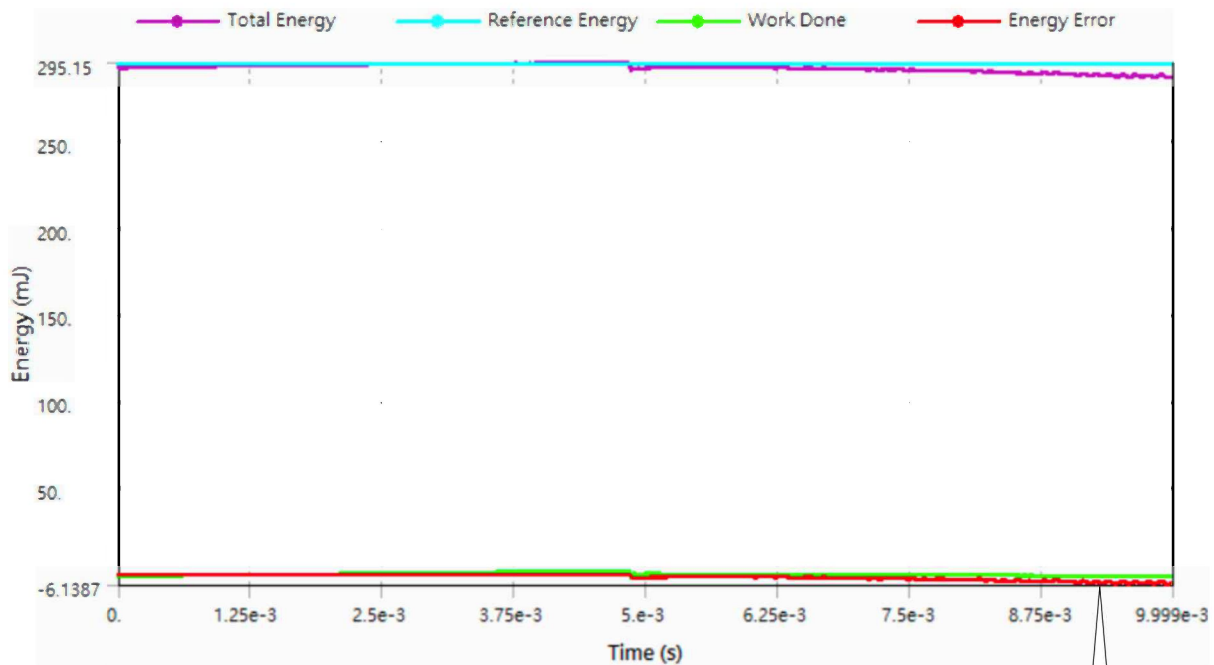
A: Explicit Dynamics
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1.e-002
Cycle Number: 80182



[5] Highlight **Total Deformation** and animate the results. ↩

Details of "Solution Information"	
Solution Information	
Solution Output	Energy Conservation
Update Interval	2.5 s
Display Points	All
Display Filter During Solve	Yes

[6] With **Solution Information** highlighted, select **Energy Conservation** in the details view. ↓



[7] The energy error (6.1387 mJ) is about 2% of the total energy (295.15 mJ), far less than the threshold. The default threshold is 10% (15.1.3[2], page 574). ↓

Wrap Up

[8] Save the project and exit Workbench. #

Section 15.4

Review

15.4.1 Keywords

Choose a letter for each keyword, from the list of descriptions

1. () Automatic Mass Scaling
2. () CFL Condition
3. () Energy Error
4. () Explicit Method
5. () Implicit Method
6. () Principle of Work and Energy
7. () Static Damping

Answers:

1. (F) 2. (E) 3. (D) 4. (B) 5. (A) 6. (C) 7. (G)

List of Descriptions

(A) A time integration method used in **Transient Structural** analysis system. It is so named because the method calculates the response in the present time using implicit information. It thus requires iterations for a time step, implying an expensive runtime for a time step. It, however, allows relatively large time steps. Overall, it is suitable for most transient simulations except high-speed or highly-nonlinear simulations.

(B) A time integration method used in **Explicit Dynamics** analysis system. It is so named because the method calculates the response in the present time using explicit information. It thus doesn't require iterations for a time step, implying an efficient runtime for a time step. It, however, requires very small time steps. Overall, it is suitable for high-speed or highly-nonlinear simulations.

(C) The principle of work and energy states that the energy at a reference time plus the work done from the reference time to a specific time is equal to the energy at that specific time.

(D) **Explicit Dynamics** uses this value to monitor the solution stability. If the energy error reaches a threshold, the solution is regarded as unstable and the computation stops. According to the principle of work and energy, the energy at a reference time plus the work done from the reference time to the present time is equal to the energy at the current time. If not equal, the difference is called an energy error. The energy error is further transformed into a dimensionless value by dividing using the maximum energy.

(E) In a single time step, a wave should not travel further than the smallest element size. This condition is used by **Explicit Dynamics** to determine the integration time step.

(F) The idea of mass scaling is to artificially increase the mass of some small elements, so that the stability time step is increased, to reduce the overall runtime.

(G) **Explicit Dynamics** is primarily for solving transient dynamic problems. However, a steady-state solution may also be obtained by introducing a damping force to critically damp the lowest mode of oscillation.

15.4.2 Additional Workbench Exercises

Performing a More Realistic Drop Test

As mentioned in 15.3.1[1] (page 587), to reduce the runtime, the model is over-simplified. In reality, the housing of a mobile phone may include a battery compartment. Include a slide-in battery in the model and perform a drop test to see if the battery will fall off upon impact with the floor.

Index

- 2D, 112
- 2D behavior, **113**, 130, 451
- 2D body, **111**, 126
- 2D graphics control, 72
- 2D model, 15, **138**, 140
- 2D simulation, 15, **109**, 112, 569
- 2D solid body, 38
- 2D solid model, **15**, 272
- 3D 20-node elements, **34**
- 3D body, **111**, 126
- 3D feature, 197
- 3D geometric modeling, 15
- 3D line modeling, 15, **272**
- 3D solid body, **37**, 68
- 3D solid element, 353
- 3D surface body, 38, **238**
- 3D solid modeling, 172
- 3D surface modeling, 15
- 3D truss, **283**, 379
- 3D view, 68
- 3D view sketching, 175
- abort a tool, 85
- ABS, 203
- ABS (PA-757), 231
- absolute displacement, 29
- Acceleration, 147
- acrylonitrile-butadiene-styrene, 203
- Activate/Deactivate at this step, 461
- active plane, 82
- active sketch, 68, **82**
- Adaptive meshing, **229**, 236
- Adaptive Multiple-Objective, 318
- Adaptive Single-Objective, **318**, 325
- Add Frozen, 128, **129**, 193
- Add Material, 129
- addendum, 97
- Adjust to Touch, 153
- advanced contact settings, 479
- All Quad, 162
- All-Hexa mesh, 228
- Allowable Change, 230
- Alternate Angle, 88
- Amplitude, 446
- analysis, 12
- Analysis Settings, 143, **460**
- Analysis Systems, 12, **112**, 426
- Analysis Type, 112
- Angle, 88
- angle unit, 71
- animate, **18**, 88
- anisotropic, 527
- ANSYS Parametric Design Language, 149
- APDL, 37, 47, **149**, 170, 239, 248
- APDL verification manual, 171
- Apply, 68
- Arc by 3 Points, 85
- Arc by Center, 73, **85**
- Arc by Tangent, 85
- arrow, 198
- Asymmetric, 562
- Augmented Lagrange, 131, 153, **478**
- Auto Constraints, **84**, 90, 107
- Auto Time Stepping, 279, 338, **491**
- automatic contact detection, 269
- Automatic Mass Scaling, **575**, 599
- averaged stress, **160**, 291
- AVI, 216
- axial load, 267
- axial stress, 142
- axis of symmetry, 125, **136**
- Axisymmetric, 130
- axisymmetric body, 270
- axisymmetric condition, 142
- axisymmetric problem, 142
- Bar, 157
- base circle, 98
- base feature, **200**, 201
- Base Plane, 120
- beam, 39
- beam bracket, **173**, 210, 249, 383
- beam element, 271, 278, **282**, 283, 310
- Beam Results/Axial Force, 292
- Beam Results/Bending Moment, 293
- Beam Results/Shear Force, 293
- Beam Results/Torsional Moment, 293
- Beam Tool, 148

- BEAM188, **39**, 283
- Bearing Load, **146**, 232, 236, 261, 265
- bearing support, 256
- beat, 414
- beat frequency, 416

- Behavior, **131**, 477
- Belleville, 537
- Belleville spring, 536
- Belleville washer, **536**, 569
- bellows joint, 239
- bending stress, 151
- beta damping, 435
- biaxial tensile test, **532**, 553, 554, 555
- Bilinear Isotropic Hardening, 579
- bisection, 481
- black arrow, 198
- Blend, 201
- Blend/Fixed Radius, 177
- blue color, **60**, 66
- bode plot, 446
- body force, 30
- Body Interactions, 582
- body types, **37**, 196
- BodyView, 130
- Bonded, 476
- Boolean, 195
- boundary condition, **17**, 47, 136
- boundary-value problems, **33**, 34Box Select, 77
- Box Zoom, 64, 72, **83**
- Bracket, **174**, 210, 249, 384
- branch, 107
- brittle material, **40**, 42, 47
- buckling, **20**, 47, 367, 368
- buckling analysis, **380**, 384
- buckling load, 365, **373**, 374, 379
- buckling mode, 367
- buckling mode shape, 378
- Building, **296**, 395, 441
- built-in unit systems, 144
- bulk modulus, 220
- calculate strain, 444
- calculate stress, 444
- Candidate Points, **320**, 321, 327
- Cantilever, 353
- Carry Over Time Step, 518
- cast iron, 40
- causes of structural nonlinearities, 470
- CD, 402
- cell, 12
- centrifugal stress, 404
- ceramics, 40
- CFL condition, **575**, 599
- Chamfer, **86**, 201
- change dimension name, 88
- change dimension value, 88
- Chart, 551
- chattering, 478
- Circle, 85
- Circle by 3 Tangents, 85
- circularity, **219**, 225
- click, 64
- click-sweep, 64
- Closed End, 71, **85**
- coefficient of thermal expansion, **32**, 283
- Coincident, 63, **89**
- compact disk, 402
- Component Systems, 57
- Components, 135
- Compression Only Support, **147**, 232, 265
- Concentric, **90**, 206
- Concept/Cross Section/I Section, 299
- Concept/Cross Section/L Section, **287**, 398
- Concept/Cross Section/Rectangular, **275**, 485
- Concept/Lines From Points, **286**, 297, 298, 398, 484
- Concept/Split Edges, **450**, 458, 483
- Concept/Surfaces From Edges, 301
- Concept/Surfaces From Sketches, **111**, 126
- conceptual model, 15
- consistent unit system, 144
- constraint status, 60, **76**, 107
- Constraints, 60, 65, 83, **89**, 319
- Construction Geometry, 164
- Construction Point at Intersection, 85
- Contact, 131, 153, **477**
- contact body, 133
- Contact BodyView, 132
- contact element, 133
- contact formulation, 477
- contact nonlinearity, 468, 470, **476**, 524
- Contact Region, **130**, 153
- contact stiffness, 525
- contact stress, 151
- Contact Tool, 148
- Contact Tool/Pressure, 507
- Contact Tool/Status, 506
- contact type, 476
- Contact/Pressure, 505
- context menu, **63**, 64, 74, 83, 107
- continuous selection, 64
- contour band, 229
- contour display, 228
- control-middle-click-drag, 64
- convergence criteria, 137, 338, **473**
- convergence study, 163, 244, 277, 282, 310, **353**, 366
- convergence value, 473
- coordinate system, **119**, 181, 224, 228, 236
- Coordinates, 152
- Coordinates File, 285
- Copies, 194

- Copy, 63, **86**
- Copy Calculated Values to Property, 557
- Copy inputs to Current, **321**, 328
- copy test data, 554
- Corner, 86
- Coulomb damping, 466
- Coulomb friction, 425
- Cover, **180**, 219, 346
- Create Coordinate System, **224**, 592
- Create Mode Shape Results, **377**, 381, 396
- Create/Body Transformation/Mirror, 105
- Create/Body Transformation/Translate, **152**, 298, 512
- Create/Boolean, 195
- Create/Pattern, **102**, 194
- Create/Primitives/Sphere, 497
- creeping, 527
- critical damping, 466
- critical damping coefficient, 422
- critical load, 373
- critically damped, 422
- cross section, 275
- Cross Section Alignments, **276**, 288, 300
- current design, **316**, 321, 322
- current energy, 574
- current sketching plane, 59
- Cursor, 84
- Curvature Normal Angle, 582
- Curve Fitting, 557
- Cut, 85, **87**
- Cut Material, **185**, 186
- cylinder cover, **179**, 219, 346
- Cylindrical, 224
- cylindrical coordinate system, **224**, 246, 542
- Cylindrical Support, 147
- D, 314
- damped free vibration, 421, **422**
- damped natural angular frequency, 422
- damping, 423
- damping coefficient, 422, 423
- Damping Controls, 425
- damping effect, 21
- damping force, 426
- damping matrix, 426
- damping mechanism, 423
- damping ratio, 422
- database, 18
- Date and Time, 229
- dedendum, 97
- Define By, 135
- definition of stress, 26
- deformed shape, 18
- degenerated element, 47
- degree of freedom, **35**, 36, 47
- Delete, 85, 111
- delete constraint, 90
- delete dimension, 89
- delete edge, 85
- Density, 403
- Depth, 68
- Design Exploration, 11, **317**
- design point, **316**, 329, 408
- design process, 311
- design space, **316**, 329
- design variable, 312, **316**
- DesignModeler, 11, **14**, 47, 57, 70
- DesignModeler GUI, 58, **81**, 197
- DesignXplorer, 312
- Details of Mesh, 16
- Details View, **60**, 66, 81
- deviatoric energy, **44**, 45
- deviatoric strain energy, 44
- deviatoric stress, **43**, 44
- Diameter, 88
- Dimension Display, 61, **62**
- dimension name, 61
- dimension value, 61, **88**
- Dimensions, 60, 66, 73, 83, **87**
- direct integration method, 426
- Direct Optimization, 317, 325
- Direction Definition, 152
- Direction of surface body, 302
- Disk, 448
- disk and block, 448
- displacement, **24**, 47
- Displacement, 146
- Displacement Convergence, **338**, 473, 524
- displacement field, **24**, 36, 160
- displacement scaling, 228
- Display, **61**, 88
- Display Model, **127**, 182
- Display Option, **158**, 159, 291
- Display Plane, **68**, 79, 100
- Display Values in Project Units, **313**, 548
- divider, 229
- Drag, 86
- Draw, 63, 83, **84**
- driving parameter, 314
- Drop, 588
- drop test, **587**, 600
- dry friction, 425
- ductile material, **40**, 42, 43, 44, 47
- Duplicate, **87**, 111, 118, 182
- Duplicate Selection, **182**, 510
- dynamic behavior, 417
- dynamic effect, 21
- dynamic simulation, **21**, 47
- earth gravity, 306
- edge, **82**, 107
- edge selection, 302
- edge selection filter, 300

- edge-to-surface contact, 264
- edges display, 228
- Edit, 88
- EFF_STN, 584
- effective strain, **583**, 584
- effective stress, 46
- Eigenvalue Buckling, **375**, 380, 384, 387
- elastic force, 426
- elastic hysteresis, 424
- elastic limit, 529
- elastic material, 424, **528**, 568
- Elastic Support, 147
- elasticity, 528
- elastomer, **10**, 11
- element, 16, **35**
- element convergence study, 170
- Element Order, **159**, 168, 355
- Element Size, **123**, 134, 154, 289, 330, 356
- element stress, 291
- element type, 37
- Ellipse, 85
- End, 64
- End Release, 283, **310**
- End selection/Place Offset, 77
- End/Set Paste Handle, 74
- End/Use Plane Origin as Handle, 63
- Energy Conservation, **574**, 598
- energy error, 574, 599
- Engineering Data, 11, **12**, 47, 211
- Engineering Data Sources, **262**, 578, 588
- Engineering Plastic, 220
- environment branch, 143
- environment condition, 16, 17, 24, 47, **145**, 170
- Equal Distance, 90
- Equal Length, 72, 76, **90**
- Equal Radius, 90
- equal temperament, 414
- equilibrium equation, **30**, 33, 34
- equilibrium iteration, 456, 466, **471**, 524
- equivalent strain, 46
- equivalent stress, **46**, 118
- Erosion Controls, 583
- ESC, **64**, 85
- Explicit Dynamics, 427, 466, **571**
- explicit integration method, 427, **572**
- explicit method, 21, 570, **573**, 599
- Export Video File, 216
- Extend, 86
- extend selection, 197, **199**
- Extend to Adjacent, 199
- Extend to Limits, **199**, 245
- Extrude, 14, 15, 68, 79, **201**
- face pair, 251
- Facet Quality, 512
- Failure, 579
- failure criterion, 19, **40**, 42, 47
- failure mode, 40
- failure point, 40
- Fatigue Tool, 148
- FE convergence, 157, **161**
- feature-based 3D modeling, 200
- FEM procedure, 35
- File/Close DesignModeler, 69
- File/Exit, **69**, 80
- File/Start Over, 83, **85**
- Fillet, 67, **86**, 177
- filleted bar, 157
- Find Face Pairs, 250
- Finger, 331
- finite element, **16**, 47
- finite element mesh, 12, **16**, 17, 47
- finite element method, 16, 18, 33, **35**
- finite element model, 12, **16**, 17, 18, 47
- first harmonic mode, 410
- first-order element, **37**, 47
- Fix Endpoints, 89
- Fix Guide Line, 193
- Fixed, 60, 66, **89**
- Fixed Support, 17, **146**
- Flexible, 99
- flexible gripper, **272**, 312
- flexible spline, 87
- Flip Horizontal, 64
- Flip Surface Normal, **301**, 302
- Flood Area, 199
- Flood Blends, 199
- Force, 146
- Force Convergence, 137, **473**, 490, 524
- force distribution, 135
- force intensity, 25
- Force Reaction, **488**, 521
- Forced Frictional Sliding, 476
- Fork, **191**, 429
- Form New Part, **303**, 399
- Formulation, 131, 153, **477**
- fracture point, 40
- fracture strength, 42
- free boundary, 47
- Free Face Mesh Type, 162
- free rotation, 198
- Free vibration, 22, 389, 417, **421**
- Frequency, **392**, 396
- Frequency Response/Deformation, 444
- Friction Coefficient, **131**, 521
- Frictional, 131, **476**, 521
- Frictionless, 476
- Frictionless Support, 125, **147**
- From Coordinates File, 285
- From Face, **185**, 188
- fundamental frequency, **393**, 426

- fundamental natural frequency, **393**, 417, 426
- GA, **272**, 281, 312
- Gear, **99**, 151
- Gearbox, 256, **257**, 390
- General, 60, 66, **87**
- General Materials, **262**, 303
- General Non-linear Materials, **578**, 588
- Generate, 68
- Generate Mesh, **114**, 133
- geometric advantage, **272**, 312
- geometric model, 13
- geometric modeling, 15
- geometric strain, **583**, 584
- Geometry, **57**, 68
- geometry display, 228
- geometry modeler, 57
- geometry nonlinearity, 23, 448, **470**, 524
- Global, 84
- global coordinate system, 173
- Global Coordinates, 224
- global mesh control, **227**, 330, 347
- governing equation, 16, 18, **30**, 34, 47
- Graph, 143
- graphic user interface, 11
- graphics window, **81**, 143
- gravitational acceleration, 25
- gravitational force, 17
- greenish-blue color, 67
- Grid, 91
- Gripper, **273**, 313
- guitar string, **410**, 456
- hardening rule, 529, **530**, 568
- harmonic mix, 413
- harmonic mode, **413**, 417
- harmonic response analysis, **427**, 441, 443, 466
- harmonic response plot, **446**, 447
- harmonic series, 413
- Henri Tresca, 42
- Hex Dominant, **340**, 351
- hexahedron, **38**, 362
- Hide Progress, 320
- Hide Sketch, 127
- high-speed impact, 467, **577**
- higher-order 2D element, 366
- higher-order element, **37**, 361
- higher-order hexahedron, 359, 361, 362, **364**
- higher-order parallel prism, **360**, 361, 363
- higher-order perpendicular prism, **360**, 361, 363
- higher-order tetrahedron, 359, 361, 362, **364**
- history of ANSYS, 51
- homogeneous, 24
- Hooke's law, 24, **31**, 32
- hoop direction, 247
- hoop stress, **142**, 286, 543, 552
- Horizontal, 63, 66, **88**, 89
- hydrostatic compressive test, 532
- Hydrostatic Pressure, 146
- hydrostatic stress, **43**, 44
- Hyperelastic Experimental Data, 554
- hyperelasticity, 19, 528, **532**, 568
- hyperelasticity model, 535
- hysteresis elasticity, 528
- idealized stress-strain curve, 529
- Ignore Axis, 65
- Impact, 578
- implicit integration method, 427, **571**
- implicit method, 21, 570, **599**
- imprint face, 498
- in-plane rotation, 278
- inconsistency, 76
- inconsistent unit system, **144**, 170
- inertia effect, 21
- inertia force, 24, **426**
- Inertial, 145, **147**
- Inertial/Acceleration, 306
- Inertial/Rotational Velocity, 405
- Inertial/Standard Earth Gravity, **306**, 594
- Initial Condition, 456, 461, **594**
- Initial Time Step, 338, **435**, 491
- initial yield point, **529**, 568
- initial yield surface, 568
- input parameter, **316**, 319, 324, 329
- insert a new object, 82
- insert APDL command, 149
- Insert as Design Point, **321**, 327
- Insert Commands, 149
- Insert/Convergence, 229
- Insert/Deformation/Total, **117**, 136
- Insert/Force, 135
- Insert/Frictionless Support, 136
- Insert/Sizing, 134
- Insert/Stress/Equivalent (von-Mises), 117
- Insert/Stress/Normal, 136
- integration time step, 575, **595**
- Interface Treatment, 153, **479**
- interpolating function, 36
- involute curve, **98**, 99
- isometric view, 68, 79, 197, **198**
- isotropic, 24
- Isotropic Elasticity, **12**, 31, 47, 211, 220
- isotropic hardening, **531**, 568
- isotropic material, 527
- iteration, 471
- Joint Load, 146
- just tuning system, 414
- kinematic hardening, **530**, 568
- Label, 148
- Lagrange multiplier, 478
- Large Deflection, **23**, 279, 338, 339, 470
- LCD, 190

- LCD display support, **203**, 231
- legend control, 229
- length unit, **58**, 71
- Length/Distance, 88
- lifting fork, 190, 429
- Line, 84
- line body, **15**, 271, 295
- Line by 2 Tangents, 84
- line model, **271**, 289
- line of action, **97**, 98
- line of centers, **97**, 98
- line of resonance, 409
- line search, **475**, 522, 524
- Linear, **159**, 355
- linear buckling analysis, 367
- linear buckling analysis with constant load, 388
- linear contact, 476
- linear elasticity, 528
- linear element, 37
- linear material, 526, **527**, 568
- linear simulation, **49**, 468, 486
- linear solution, **337**, 494
- linear structure, 468, **524**
- Linearized Stress, 148
- linearly elastic, **24**, 31
- liquid crystal display, 190
- Load Multiplier, **377**, 382, 386
- load step, 456, **471**, 524
- Loads, 17, 145, **146**
- Loads/Bearing Load, 265
- Loads/Force, 213
- Loads/Moment, 155
- Loads/Pressure, 305
- local coordinate system, **175**, 274
- lofting guide line, 193
- Look At, 59, 64, **72**, 182
- Lower Bound, **319**, 326
- lower-order element, **37**, 361
- lower-order hexahedron, 355, 361, 362, **364**
- lower-order parallel prism, **361**, 363
- lower-order perpendicular prism, **361**, 363
- lower-order prism, 357, 358, **364**
- lower-order tetrahedron, 356, 361, 362, **364**
- lower-order triangular element, 159
- lower/upper bound, 319
- lumped mass model, **420**, 466
- M20x2.5, 92
- magnitude, 145
- major diameter, 92
- Major Grid Spacing, 91
- Manual Input, 296
- Manual Source, 357
- Mass, 324
- Mass Coefficient, 425
- mass density, 25
- mass matrix, 426
- material assignment, 113, **212**
- material damping, **424**, 466
- material model, 19, **31**
- material nonlinearity, 468, **470**, 524
- material parameter, **31**, 526
- material property, 220
- material symmetry, 32
- Max Modes to Find, **376**, 380, 391
- Max Refinement Loops, 230
- Maximum Energy Error, 574
- Maximum Equivalent Plastic Strain EPS, 579
- maximum normal stress, 41
- maximum principal stress, 41
- maximum shear stress, 41
- maximum shear stress criterion, 42
- Maximum Time Step, **338**, 491
- measuring damping coefficient, 422
- Mechanical, 11, **16**, 49
- Mechanical GUI, 16, 18, 113, **142**
- mechanical properties of PDMS, 11
- medium principal stress, **41**, 43
- member force, 293
- member moment, 293
- Mesh, 114
- mesh control, 159
- Mesh Control/Face Meshing, 431
- Mesh Control/Method, 217
- Mesh Control/Sizing, **167**, 514
- mesh count, 227
- mesh density, **218**, 347
- Mesh Metric, **335**, 347
- mesh quality, **335**, 336
- Mesh/Generate Mesh, **133**, 135
- Mesh/Preview Surface Mesh, **501**, 502
- meshing, 330
- Method, 159
- Method Name, 325
- Microgripper, 103, **104**, 495
- Micrometer, 104
- Mid-Surface, 249, **251**
- middle mouse button, 198
- middle-click-drag, 64
- Midpoint, 89
- midside node, 37
- mild steel, 40
- minimum normal stress, 41
- minimum principal stress, **41**, 383
- Minimum Reference, 473
- Minimum Time Step, **338**, 491, 519
- Min Size Limit, 349
- Minor-Steps per Major, 91
- Mirror, 105
- MISQP, 318
- Modal, 390

- modal analysis, 22, 49, 389, 381, 391, **426**
- Mode, **377**, 381
- mode shape, 426
- mode tab, 81
- Model, 16
- model airplane wing, 418
- model tree, 68, 79, **82**, 107
- Modeling mode, **68**, 81, 107
- Modify, 63, 65, 67, 83, **86**, 118
- MOGA, 318
- Mohr's circle, 27, **41**, 42
- Moment, 146
- Moment Convergence, 473, **490**, 524
- moment equilibrium, 27
- Mooney-Rivlin, **535**, 553
- Mooney-Rivlin 2 Parameter, 557
- mouse cursor, 198, **199**
- mouse operation, 64
- Move, **87**, 88
- move dimension, 67
- MPC, 477
- multi-point constraint, 477
- Multilinear Isotropic Hardening, 548
- MultiZone, 217, **227**
- MultiZone method, 236, **341**, 352
- MythBusters, 402
- Name, 61
- natural angular frequency, 421
- natural frequency, 22, 389, **393**, 406, 407, 417, 421, 426
- natural period, 421
- negative-X-face, 26, **27**
- negative-Y-face, 26, **27**
- negative-Z-face, 26, **27**
- Neo-Hookean, 535
- New Chart and Table, **547**, 567
- New Plane, **82**, 120, 181
- New Section Plane, 343
- New Sketch, **82**, 106, 127, 186
- Newton-Raphson method, **472**, 524
- Next View, 72
- NLPQL, **318**, 329
- No Separation, 466, **476**
- nodal displacement, 36
- nodal stress, 291
- node, 16, 35, **49**
- nonlinear buckling analysis, 367, **387**
- nonlinear contact, 476
- Nonlinear Effects, 212
- nonlinear elastic material, 528
- nonlinear elasticity, 528
- nonlinear material, 526, **527**, 568
- nonlinear simulation, 49, 137, 337, 468, **469**, 489
- nonlinear solution, 338, 344, **494**
- nonlinear structure, 468, **524**
- nonlinearity, 264
- Normal Lagrange, 478
- Normal Stiffness, 477, **478**, 525
- normal strain, 28, **29**
- normal stress, **26**, 27
- number of buckling modes, 376
- number of digits, 229
- Number of Steps, **471**, 515
- Number Of Substeps, 279
- numerical method, 33
- object, 107
- objective, 319
- Objectives and Constraints, **319**, 325
- octave, 414
- Offset, 77, **87**, 183, 194
- Ogden form, 535
- Open End, 63, **85**, 99
- Open End with Fit Points, 99
- Operation, 128
- optimal design, **320**, 321, 327
- Optimization, 311, **317**, 318, 320, 325
- optimization method, 318
- optimization problem, 325
- order of element, 36
- Orientation, 136
- Orientation About Principal Axis, 228
- Origin, 228
- orthotropic, 527
- Orthotropic Elasticity, 32, 527
- out-of-plane translation, 278
- Outline, **143**, 316
- Outline Plane, 181, **182**
- Output Controls, **435**, 453, 519
- output parameter, 316, 324, **329**
- Oval, 85
- over-constrained, 76
- over-damped, 422
- overtone, 413
- P, 315
- paint-select, 77
- Pan, 64, **72**, 83
- Parallel, **90**, 558
- parallel prism, **357**, 363
- Parameter Set, **314**, 315
- parameter study, 373
- part, 82
- pascal, 31
- Paste, 64, **87**
- Paste at Plane Origin, **64**, 75
- paste handle, **75**, 107, 127
- patch, 348
- Patch Conforming, 348
- Patch Independent, **348**, 349
- Path, **164**, 165
- PC, 402
- PDMS, 10, **11**

- perfectly elastic-plastic material, 577
- Perpendicular, 89
- perpendicular prism, **358**, 363
- Physical Properties, 220
- physics of music, 410
- pin-jointed, 310
- Pinball Region, 479
- Pitch, 198
- pitch circle, **97**, 98
- pitch point, **97**, 98, 99
- pitch radius, **97**, **98**
- placed feature, 200, **201**
- planar seal, 553
- plane of symmetry, **17**, 125
- plane outline, 182
- Plane Strain, 561
- Plane Stress, 110, **113**, 513, 451
- plane view sketching, 175
- plane-strain condition, 141
- plane-strain problem, **141**, 170, 553
- plane-stress condition, 140
- plane-stress problem, **140**, 170
- PLANE182, 38
- PLANE183, 38
- plastic material, **528**, 548, 568
- plastic strain, **528**, 577, 583
- Plastic Strain Failure, 579
- plasticity, 528, **529**, 536
- plasticity model, 531
- Play, **18**, 216
- pneumatic finger, **10**, 35, 331, 569
- Point, **73**, 74, 82
- Point, 296
- Poisson's ratio, 11, **31**, 32
- Poisson's Ratio, 12, **220**
- polycarbonate, 402
- polydimethylsiloxane, 10
- Polygon, 84
- Polyline, 63, **84**
- polynomial form, 535
- polyoxymethylene, **272**, 481
- POM, **272**, 481
- Preserve Bodies, 152
- Pressure, 25, **146**
- pressure angle, 97
- pressure cylinder, **179**, 219, 346
- prestressed modal analysis, 390, **393**
- pretension, 125
- Previous View, 72
- Principal Axis, 228
- principal direction, 41
- principal strain, 19
- principal stress, **41**, 49
- principal stretch ratio, 534
- principal view, 197
- principle of conservation of energy, 574
- principle of work and energy, **574**, 599
- Probe, 148
- Probe/Deformation, **437**, 462
- Probe/Force Reaction, **488**, 516, 544, 566
- Probe/Stress, 439
- problem domain, 24
- procedure of FEM, 49
- Progress, 320
- Project, 211
- project name, **69**, 80
- Project Schematic, 11, 12, 49
- project tree, **143**, 170
- project unit system, 144
- Properties, 112
- Pull-down menu, **81**, 143
- Pure Penalty, 478
- quadratic element, 37
- quadrilateral, 38
- Quadrilateral Dominant, 162
- quadrilateral element, **37**, 162, 163
- quadrilateral-based pyramid, 38
- quality of mesh, 16
- quasistatic simulation, 419
- quadrilateral shell, 38
- radial deformation, 247
- radial direction, 246
- radial stress, 142
- Radius, 67, 86, **88**
- random vibration analysis, **428**, 466
- rate-dependent material, 568
- Re-Fit Spline, 99
- reaction force, 488
- Rectangle, 59
- Rectangle by 3 Points, 85
- Redo, 72
- reference energy, 574
- Refresh Geometry, 123
- Remote Displacement, 147, **504**
- Remote Force, 146
- Rename, 137
- Rename Based on Definition, 385
- Replicate, 74, **87**
- Reset, 112, **346**
- reset legend, 229
- residual force, **472**, 524
- residual strain, 528
- resolution, 214, **227**
- response, **24**, 30
- response frequency, 435, **437**
- response spectrum analysis, **428**, 466
- restart, 595
- Result Sets, **280**, 438
- results, 18

- results object, 117, **148**
- results tool, 148
- results toolbar, 148
- results view control, 228
- Resume From Cycle, 596
- Retrieve This Result, **541**, 551, 552
- Reverse Normal/Z-Axis, 120
- revolute joint, 283
- Revolve, 96, **201**, 242
- Richard von Mises, 44
- right-click, 64
- right-click-drag, 64
- Rigid, **581**, 592
- rigid body, 499
- rigid body mode, 370, **393**, 417
- rigid body motion, 29, **117**, 118
- rigid-jointed, 310
- roll, 198
- rotate, 64, 197, **198**
- Rotate about Global Z, 120
- Rotate by r, **74**, 75
- Rotation Convergence, **473**, 524
- rotation mode, 412
- Rotational Velocity, **147**, 405
- Rough, 476
- round, 177
- round-cornered textbox, 58
- RPM, 408
- rubber, 553
- ruler, **61**, 113
- Safety Factor, 215
- Save Project, **69**, 80
- Scoping Method, **165**, 505
- Screening, 318
- scroll-wheel, 64
- SDOF, 421
- Seal, 554
- second-order element, **37**, 49
- section plane, 343
- section view, 125, **343**
- Select All, 245
- select mode, 77
- Select new symmetry axis, **60**, 65
- selection, 199
- Selection Filter, 74, 107, 197, **199**
- selection pane, 197, **199**, 301, 302
- Semi-Automatic, 88
- separator, **81**, 143, 211
- Settings, 83, **91**
- shape function, **36**, 49
- shape memory alloy, **103**, 495
- sharp-cornered textbox, 58
- shear failure, **40**, 42
- Shear Modulus, 31, 32, **220**
- shear strain, **28**, 29
- shear strength, 42
- shear stress, **26**, 27
- Shear Test Data, 532, 553, **554**, 555
- shell element, 238, 269, **270**, 278
- SHELL181, 38
- SHELL208, 248
- Shift-middle-click-drag, 64
- Show Constraints, 90
- Show Elements, 307
- Show in 2D, 91
- Show Progress, 320
- Show Sketch, 128
- Show Whole Elements, 343
- Show/Sweepable Bodies, 341
- SimpleBeam, 369
- simply supported beam, 368
- simulation, 12
- single degree of freedom model, 421
- singular point, 169
- singular stress, 166
- sizing, **154**, 330
- sketch, **82**, 107
- sketched feature, 200
- Sketching, 15, 56, **62**
- Sketching mode, 59, **81**, 107
- sketching option, 61
- sketching plane, **82**, 107
- sketching toolboxes, 81, **83**
- Skewness, **335**, 347
- Skin/Loft, **192**, 201
- SMA, **103**, 495
- small cube, 26
- Small deformation, 24, **28**, 29, 31
- Smooth Contours, 586
- Snap, 91, **509**
- Snap lock, 508
- Snaps per Minor, 91
- solid body, 15
- solid damping, 424
- solid element, 270
- Solid Fill, 463
- solid model, 255
- SOLID186, 37
- SOLID187, 37
- Solution, 117, **143**
- solution accuracy, 575
- Solution Information, 137, 143, 338, **474**, 490
- Solution Output, 137
- Solve, **18**, 116
- Solve Curve Fit, 557
- Solver Output, **474**, 490
- source face, **228**, 341
- Sphere Radius, **167**, 168
- Spline, **85**, 99
- Spline Edit, **87**, 99

- Spline Edit option, 87
- Split, 86
- Split Edges, 450
- Split option, 86
- spur gear, **97**, 151
- square bracket, 10
- stability analysis, 367
- Standard Earth Gravity, 147
- standing wave, 410
- Static Damping, **576**, 599
- static simulation, 21
- Static Structural, 11, 112
- statically indeterminate structure, 379
- Statistics, 16, **114**
- status bar, **81**, 83, 143
- status symbol, 149
- steady state, 419
- Step End Time, 435
- Steps, 456, 466, **471**
- stiffness, 49
- Stiffness Behavior, 581
- Stiffness Coefficient, **425**, 435
- stiffness matrix, **36**, 49, 426
- still water, 25
- Stop, 216
- strain, **28**, 49
- strain component, 28
- strain energy, **533**, 534
- strain field, 36
- strain invariant, 534
- strain state, 49, **141**, 142
- strain-displacement relation, **31**, 33, 36
- stress, **25**, 49
- stress component, **26**, 27
- stress concentration, 163
- stress discontinuity, **157**, 170
- stress field, 36, **160**
- stress intensity, **42**, 49
- stress singularity, 157, **166**, 169, 170
- stress state, **27**, 41, 42, 46, 49, 140, 142
- stress stiffening, 20, 49, 367, **368**
- stress stiffening effect, **373**, 387
- Stress Tool, 148
- stress-strain curve of PDMS, 11
- stress-strain relation, **31**, 33, 36
- Stress/Error, 161
- Stress/Maximum Bending Stress, 291
- Stress/Minimum Bending Stress, 291
- stretching mode, 412
- String, **410**, 457
- strong dimension, 79
- structural analysis problem, 24
- structural dynamic analysis, 417
- structural dynamics, 420
- Structural Error, 157, **161**, 170, 216
- structural mechanics, 24
- structural nonlinearity, 22
- structural steel, 113
- SU316, 239
- subsequent yield point, **529**, 568
- Substeps, 456, 466, **471**, 524
- Subtype, 182
- Support, 17, 136, 145, **146**, 204, 231
- support condition, 265
- supporting file, 69
- Supports/Fixed Support, 212
- surface body, 15, 113, 242, 257, **269**, 295, 301
- surface force, 30
- surface model, 238, **255**, 289
- surgical parallel robot system, 10
- Sweep, 192, **201**
- Sweep method, 236, **341**
- sweep-select, 77
- sweepable, 228
- sweepable body, 341
- switch between steps, 471
- Symmetric, **131**, 477
- Symmetry, **60**, 65, 76, 89, 118, 270
- symmetry condition, 278
- symmetry of shear stress, 27
- Table of Design Points, **316**, 317
- Table of Outline, 326
- Table of Schematic, **319**, 320
- Tabular Data, **143**, 461, 518, 540
- tag, 228
- Tangent, **89**, 106
- Tangent Line, 84
- Tangent Modulus, 579
- tangent stiffness, 472
- tangential stiffness, 478
- Target, 131, 153, **477**
- Target BodyView, 132
- target face, **228**, 341
- Temperature, 555
- temperature change, **17**, 24
- tensile failure, **40**, 42
- test data, 532
- tetrahedron, **38**, 362
- Tetrahedrons method, 356
- textbox, 10
- Thermal Condition, 146
- thermal effects, 32
- thermal strain, 32
- Thermal Strain Effects, 212
- Thickness, **113**, 244
- Thickness assignment, 253
- thin body, 206
- thin solid body, 257, **269**
- Thin/Surface, **201**, 206, 256, 257

- threaded bolt, 92
- threaded bolt-and-nut, 125
- Threads, **93**, 126
- threshold, 250
- thunderbolt, 121
- Time, 338
- Time Integration, 457
- Time Step, 471
- Time Step Controls, 479
- Time Step Safety Factor, 575
- time-dependent stress-strain relation, **527**, 568
- Tolerance, 473
- toolbar, **81**, 143
- toolbar menu, 115
- Toolbox, **11**, 12, 316
- Tools, 148
- Tools/Beam Tool, **279**, 291
- Tools/Contact Tool, 505
- Tools/Mid-Surface, **250**, 251
- Tools/Options, 62
- Tools/Stress Tool, 213
- Tools/Symmetry, 119, 120
- topology nonlinearity, 468, **470**
- torsional vibration, 397
- total strain energy, 44
- Transfer Data To New/Eigenvalue Buckling, 374
- transformation, 181
- transient state, 419
- transient structural simulation, 21, 417, **419**, 426, 434
- Transient/Initial Conditions, 453
- Transjoint, 482
- translational joint, 481
- Tree Outline, 81
- Tresca criterion, 42
- Triad, 197
- triangle, 38, **159**, 364
- triangle-based prism, 38
- triangular element, **162**, 163
- triangular shell, 38
- Trim, 65, **86**
- Triplate, **80**, 110, 323
- True Scale, **156**, 247, 438
- Truss, **284**, 380
- twelve-tone equally tempered tuning system, 414
- two-step method, 456
- two-story building, **295**, 395, 418, 441
- Type, 521
- type of coordinate system, 228
- U.S. customary unit system, 151
- Unaveraged, **158**, 159
- unaveraged stress, **160**, 291
- undamped free vibration, 421
- undeformed shape, 10
- under-constrained, **67**, 76
- Undo, 72
- uniaxial compressive test, 532
- uniaxial tensile test, 19, **40**, 42, 532, 553, 554, 555
- uniform velocity, 448
- unit system, 144
- Unit Systems Mismatch, 445
- Unite, 195
- Units, 58, 71, 113, **144**
- Units/Display Values as Defined, 408
- Units/RPM, 405
- unprestressed modal analysis, 390
- Update, 320
- Update All Design Points, 317
- Update Selected Design Points, **317**, 321, 328
- Update Stiffness, 479
- Upper Bound, **319**, 326
- Use Manufacturable Values, 326
- Use Maximum, 444
- Use Plane Origin as Handle, 75
- User Defined Result, **148**, 546, 584
- UY, 546
- Value, 61, **62**, 473
- vector display, 228
- Velocity, 453
- verification manual, 525
- verification manual for Workbench, 237
- Vertical, 63, **88**, 89
- vibration mode, 22
- vibration mode shape, 389, **393**, 397
- view results, 148
- view rotation, 198
- View/Cross Section Alignments, **274**, 286, 288
- View/Cross Section Solids, **275**, 288, 300, 486
- View/Reset Workspace, 211, **315**
- View/Ruler, **61**, 113
- View/Shaded Exterior, 178
- View/Thick Shells and Beams, **254**, 412, 487
- View/Windows/Reset Layout, 142
- View/Windows/Section Planes, 343
- viscosity, 527
- viscous damping, **423**, 466
- VM25, 171
- volumetric test, 532
- von Mises strain, 46
- von Mises stress, 19, 20, **46**, 49
- von Mises yield criterion, **43**, 44, 46, 529
- von Mises yield surface, **46**, 568
- W16x50, 69
- W16x50 beam, **57**, 295, 299
- wave speed, 410
- weak constraint, 65
- weak dimension, **67**, 78, 86
- Weak Springs, **116**, 170
- well-defined, **60**, 66, 76
- Workbench GUI, **11**, 49, 57
- Worksheet, **148**, 505

X-face, 26
XYPlane, **14**, 59
Y-face, 26
Yeoh, 535
yield criterion, 529
yield point, 40
yield state, 529
yield strength, 211
yield surface, 568
young's modulus, **11**, 31, 32, 529
Young's Modulus, **12**, 220
Z-face, 26
Zoom, 72
zoom in/out, 64
Zoom to Fit, 60, 64, 67, 68, **72**

Finite Element Simulations with ANSYS® Workbench 2021

Theory, Applications, Case Studies

- A comprehensive easy to understand workbook using step-by-step instructions
- Designed as a textbook for undergraduate and graduate students
- Relevant background knowledge is reviewed whenever necessary
- Twenty seven real world case studies are used to give readers hands-on experience
- Comes with video demonstrations of all 45 exercises
- Compatible with ANSYS Student 2021
- Printed in full color

Description

Finite Element Simulations with ANSYS Workbench 2021 is a comprehensive and easy to understand workbook. Printed in full color, it utilizes rich graphics and step-by-step instructions to guide you through learning how to perform finite element simulations using ANSYS Workbench.

Twenty seven real world case studies are used throughout the book. Many of these case studies are industrial or research projects that you build from scratch. Prebuilt project files are available for download should you run into any problems. Companion videos, that demonstrate exactly how to perform each tutorial, are also available.

Relevant background knowledge is reviewed whenever necessary. To be efficient, the review is conceptual rather than mathematical. Key concepts are inserted whenever appropriate and summarized at the end of each chapter. Additional exercises or extension research problems are provided as homework at the end of each chapter.

A learning approach emphasizing hands-on experiences is utilized though this entire book. A typical chapter consists of six sections. The first two provide two step-by-step examples. The third section tries to complement the exercises by providing a more systematic view of the chapter subject. The following two sections provide more exercises. The final section provides review problems.

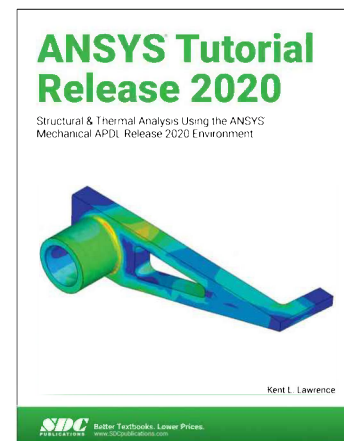
ANSYS Student 2021

ANSYS provides a free product license for students anywhere in the world. The only limitation is the problem size. All examples in this book are designed to meet this limitation and have been tested with the free student version.

Table of Contents

1. Introduction
 2. Sketching
 3. 2D Simulations
 4. 3D Solid Modeling
 5. 3D Simulations
 6. Surface Models
 7. Line Models
 8. Optimization
 9. Meshing
 10. Buckling and Stress Stiffening
 11. Modal Analysis
 12. Transient Structural Simulations
 13. Nonlinear Simulations
 14. Nonlinear Materials
 15. Explicit Dynamics
- Index

Also Available



Better Textbooks. Lower Prices.
www.SDCpublications.com



ACCESS CODE
UNIQUE CODE INSIDE

SUGGESTED PRICE
Retail \$84
School Bookstores \$53

@Seismicisolation
@Seismicisolation

READER LEVEL
Beginner to Intermediate

Not returnable if code is redeemed

ISBN 978-1-63057-456-7



9 781630 574567